72nd Annual Meeting of the APS Division of Fluid Dynamics
Seattle, Washington
https://www.aps.org/meetings/meeting.cfm?name=DFD19
3:00PM A01.00001 Numerical Predictions of Spray Root Location, Hydrodynamic Pressure, and Structural Deformation of a Highly Flexible Deforming Wedge During Vertical Water Entry  
John Gilbert, Christine Gilbert, Virginia Polytechnic Institute and State University — Wave interactions with high-speed planing craft result in repeated slamming events that can lead to equipment failure and serious injury. Our earlier efforts investigated this problem through the use of wedge drop experiments and theoretical predictions, focusing on the effects of linear-elastic structural response on spray root propagation and hydrodynamic pressure. In the current work, predictions from a coupled, non-linear finite element - non-linear boundary value approach are used to extend our previous study to include the effects of large structural deformation. Observations from recent flexible wedge drop experiments conducted at the Virginia Tech Hydroelasticity Laboratory are used to inform the development of the solver. Numerical predictions for spray root location and hydrodynamic pressure as a function of time show good agreement with experiments, while predictions for the structural deformation show discrepancies. This work is part of a larger research effort aimed at understanding this fluid-structure interaction problem.

1This project is funded by ONR grant number: N00014-16-1-3188.

3:13PM A01.00002 Effect of convexity & texture on the water entry of a cone; inspired by diving birds  
Lianxia Li, Mark Fenn, Maysam Mousaviraad, University of Wyoming, Christine Gilbert, Virginia Tech — Fluid-structure interaction (FSI) problems are complex in nature, especially when the loading by the fluid is unpredictable and impulsive or when the material properties of the structure are nonlinear or otherwise complex. The purpose of this study is to investigate the water impact hydrodynamics/hydro-elasticity of flexible bodies through computational and experimental methods. Computational FSI capability is developed at University of Wyoming based on a two-way coupled approach. The flow is resolved by URANS CFD modeling and non-linear FEM equations are employed for structural dynamics. Slaming experiments are carried out at Virginia Tech for a 20 degree angle aluminum wedge with a bottom plate thickness of 3.175 mm and drop heights ranging from 0.9 to 38.1 cm. Measurements include time histories of vertical position, acceleration, pressure, and strain. Strain measurements are taken at single points using strain gauges as well as full-field measurements of all strain components and out-of-plane deflection using stereoscopic digital image correlation (S-DIC). Computational studies include validation and investigations of the physics involved in the interactions. Next step will extend to highly flexible structures where the two-way nonlinear interactions are more significant.

3:26PM A01.00003 Fluid-structure interaction (FSI) of Wedge Drop Slamming  
Lianxia Li, Mark Fenn, Maysam Mousaviraad, University of Wyoming, Christine Gilbert, Virginia Tech — Fluid-structure interaction (FSI) of flexible bodies through computational and experimental methods. Computational FSI capability is developed at University of Wyoming based on a two-way coupled approach. The flow is resolved by URANS CFD modeling and non-linear FEM equations are employed for structural dynamics. Slamming experiments are carried out at Virginia Tech for a 20 degree angle aluminum wedge with a bottom plate thickness of 3.175 mm and drop heights ranging from 0.9 to 38.1 cm. Measurements include time histories of vertical position, acceleration, pressure, and strain. Strain measurements are taken at single points using strain gauges as well as full-field measurements of all strain components and out-of-plane deflection using stereoscopic digital image correlation (S-DIC). Computational studies include validation and investigations of the physics involved in the interactions. Next step will extend to highly flexible structures where the two-way nonlinear interactions are more significant.

3:39PM A01.00004 How air deforms the free surface just before disk impact on a liquid bath  
Devaraj Van Der Meer, Utkarsh Jain, Anaïs Gautier, Detlef Lohe, University of Twente — When a flat disk impacts onto a liquid bath, a layer of air is trapped between the disk and the free surface, a phenomenon known as air cushioning. The air layer is pushed out at increasing speeds, causing the water surface to be lifted up towards the approaching disk. This qualitative observation is traditionally ascribed to Bernoulli suction occurring in the low-pressure region created by the large air velocities in the gap. Here, by means of a novel high-speed imaging technique that uses the free surface as a mirror, we quantitatively measure the time evolution of the free surface profile. The predicted elevation of the free surface below the disk is observed to be followed by an unanticipated depression just outside the disk’s edge. Although this depression starts growing at a later point in time, its magnitude eventually surpasses that of the elevation. We show that the results are inconsistent with Bernoulli suction and that instead, the deformation appears to be initiated by a Kelvin-Helmholtz instability occurring under the edge of the approaching disk.

3:52PM A01.00005 Passive Cavity Deflation after Water Entry  
Emma Fraley, Rafsan Rabb, Utah State University, Kelli Hendrickson, Massachusetts Institute of Technology, Tadd Truscott, Utah State University — Under specific impact conditions, objects form cavities when they enter a quiescent surface and persist for an extended time. The ability to quickly deflate or remove the cavity is important for naval, acoustic and marine applications. Herein, we propose a passive cavity deflation mechanism made of a hollow tube attached to a sphere with radial holes near the base, called the Kiara tube. We perform a series of experiments varying tube length, radial hole area and drop height to assess what factors influence cavity deflation and utilize high-speed cameras and image processing to quantify the cavity deflation. In general, tube lengths less than a critical length provide no deflation while larger tube lengths provide a consistent deflation rate. We will also discuss the effects of additional factors such as drop height, radial hole area, surface coatings, and forces as they relate to the cavity deflation rate.
The effect of bending stiffness on the impact of flexible plates on a water surface\textsuperscript{1}, AN WANG, KIT PAN WONG, MIAO YU, KENNETH T. KIGER, JAMES H. DUNCAN, University of Maryland, College Park, UNIVERSITY OF MARYLAND, COLLEGE PARK TEAM — The controlled oblique impact of a series of flexible/rigid rectangular plates (length 108 cm, width 41 cm, pitch angle 10\textdegree) on a quiescent water surface is studied experimentally. The flexible plates, which are made of 6061 aluminum, have various thicknesses while the rigid plate structure is made of two aluminum plates stiffened by a light-weight frame glued between them. The mounting structure allows each end of the plate to rotate around a transverse horizontal axis located slightly above the plate. For each plate, the motion trajectory and the impact speed are varied. The plate thickness and impact speed are chosen so that the impact time scale is on the same order of the wet natural period of the plate. The spray formation, the transient impact force, the moment about a horizontal transverse axis, the deflection along the plate’s center line and the impact pressure on the plate’s lower surface are measured with various techniques. For the same plate, it is found the maximum impact force/moment scale with the square of the component of the impact velocity normal to the undeformed plate. The temporal evolution of impact force and moment arm each have two distinct slopes. The relationship of this behavior to the plate’s deformation is explored.

\textsuperscript{1}The support of the Office of Naval Research under grants N000141612619 and N000141612748 is gratefully acknowledged.

Tuning the restitution coefficient of closed containers partially filled with water\textsuperscript{1}, PABLO GUTIERREZ, GUSTAVO CASTILLO, Universidad de O’Higgins, VICENTE SALINAS, Universidad Autónoma de Chile, FRANCISCO OLEA, LEONARDO GORDILLO, Universidad de Santiago — It is well known that when open containers partially filled with water fall, a strong jet emerges after they hit the ground (Milgram, JFM, 1969). Surprisingly Antkowiak et al. (JFM, 2007) showed that if the water surface of the flow remains perfectly flat during the fall, no jet forms at all. In this work, we show how the shape of the water surface just before the impact not only has an effect on the presence of the jet but also on the rebound of the overall system (container+water). We focus in closed containers and study the influence of the filling fraction, which has been shown to be a relevant ingredient in the bouncing of spherical containers filled with grains and liquid. Our observations suggest a different mechanism of why the flat surface prevents jet formation compared to the open container and the viscous sublayer is not the key ingredient in the jet formation. From the overall system energy point of view, the reflection coefficient is always higher than the reflection coefficients of the container and liquid interfaces taken separately. We hypothesize that the two interfaces act together to generate the final energy reflection coefficient.

\textsuperscript{1}Swedish Research Council Grant No. 63820139243

A theory for the Pierson and Moskowitz spectrum\textsuperscript{1}, ANTHONY BONFILS, WOOSOK MOON, DHUBADITYA MITRA, JOHN WETTLAUFER\textsuperscript{2}, Nordic Institute for Theoretical Physics — The spectrum of surface gravity waves was measured in the sixties by Pierson and Moskowitz for a fully developed sea, however a theory explaining their observations has been lacking. In 1957, Miles proposed an instability mechanism for single mode growth involving the solution of the hydrodynamic equation with high growth at the critical level, which is the height at which the phase speed of the wave is equal to the wind speed. The turbulent wind is modeled by a logarithmic profile. We show that the introduction of a viscous sublayer does not affect the growth rate. Because there is a continuous spectrum of waves, we study the interaction between the different critical layers (the counterpart of wave-wave interactions) in the framework of the quasi-linear model of Janssen (1982). The natural small parameter in this system is the air/water density ratio, which introduces a scale separation in wind-wave generation. Whereas the evolution of gravity waves occurs on a short (fast) time scale, the energy transfer from the wind occurs on a long (slow) time scale. The feedback of growing surface waves on the wind profile occurs on an intermediate time scale. A decrease of its curvature leads to the saturation of the waves, which is the first step in the transition towards a fully developed sea.

\textsuperscript{1}Nordic Institute for Theoretical Physics — The fundamental knowledge of the airflow over the surface of the ocean is essential for evaluating air-sea fluxes. Despite recent numerical/theoretical advances, experimental data near the water interface have been difficult to obtain, especially for scales at which viscosity plays a role. Here, we present direct measurements of the velocity field in the turbulent airflow above wind waves for a range of 10-m wind speed varying from 2 to 17 m/s. Improvements in measuring techniques have allowed us to detect the viscous sublayer in the airflow near the interface and make direct measurements of the airside viscous stresses. In a phase-averaged sense, the viscous stress is highest on the upwind face of wave crest with its peak value close to the wave crest and its minimum about the middle of the leeward side of waves. At wind speeds of 2 m/s, corresponding to winds in which wind waves are first generated, the mean tangential stress represents more than 50\% of the wind stress. The contribution of the viscous stress to the total momentum flux decreases significantly with increasing wind speed. In low winds, we also observe the viscous stress generated by the wave motion in the airflow. To the best of our knowledge, these are the first measurements of airside, wave-induced viscous stresses.

\textsuperscript{1}This research was supported by the National Science Foundation (NSF) through grant numbers OCE-0748767, OCE-1458977, and OCE-1634051.

Wave- and shear-induced viscous stress over wind waves\textsuperscript{1}, KIANOOSH YOUSEFI, FABRICE VERON, School of Marine Science and Policy, University of Delaware, MARC BUCKLEY, Institute of Coastal Research, Helmholtz-Zentrum Geesthacht — Detailed knowledge of the airflow over the surface of the ocean is essential for evaluating air-sea fluxes. Despite recent numerical/theoretical advances, experimental data near the water interface have been difficult to obtain, especially for scales at which viscosity plays a role. Here, we present direct measurements of the velocity field in the turbulent airflow above wind waves for a range of 10-m wind speed varying from 2 to 17 m/s. Improvements in measuring techniques have allowed us to detect the viscous sublayer in the airflow near the interface and make direct measurements of the airside viscous stresses. In a phase-averaged sense, the viscous stress is highest on the upwind face of wave crest with its peak value close to the wave crest and its minimum about the middle of the leeward side of waves. At wind speeds of 2 m/s, corresponding to winds in which wind waves are first generated, the mean tangential stress represents more than 50\% of the wind stress. The contribution of the viscous stress to the total momentum flux decreases significantly with increasing wind speed. In low winds, we also observe the viscous stress generated by the wave motion in the airflow. To the best of our knowledge, these are the first measurements of airside, wave-induced viscous stresses.

\textsuperscript{1}This research was supported by the National Science Foundation (NSF) through grant numbers OCE-0748767, OCE-1458977, and OCE-1634051.

Observations of water waves and wind-wave interactions in the Gulf of Aqaba (Eilat)\textsuperscript{1}, ALMOG SHANI-ZERBIB, RONI HILEL GOLDSHIMID, DAN LIBERZON, Technion - Israel Institute of Technology, T-FAIL TEAM — The Gulf of Aqaba (Eilat) at the northern tip of the Red Sea has a unique elongated rectangular shape, a stable diurnal cycle of wind regime and a steady climate. The resulted wind induced wave field is heavily influenced by the topography and bathymetry of the gulf area. These conditions make the gulf area an alluring location for wind-wave interactions research. Here, we report on a first ever high-resolution observations of the water wave regimes in the Gulf of Aqaba, conducted during two short campaigns in 2017 and 2019. These include wave field measurements using a directional wave gauge array accompanied by wind velocity field measurements at several heights. High resolution measurements of the wind flow turbulent characteristics were also conducted using the newly developed combo probe, a collocated ultrasonic and hot-wire anemometer, utilizing an in-situ Neural Network calibration procedure. We will report on the characteristics of the wind induced water wave fields and detail the wind-wave interaction dependence on the variations of the wind flow turbulence characteristics in terms of mean and fluctuating values. Empirical fits of the turbulent flow length scales vs. the wave field characteristics will also be presented.

\textsuperscript{1}ISF Grant 1521/15
3:39PM A02.00004 Influence of the free surface on the distribution of buoyant particles in wavy flow
- MICHELLE DIBENEDETTO, Woods Hole Oceanographic Institution.
- JEFFREY KOSEFF, NICHOLAS OUELLETTE, Stanford University — Using numerical and analytical techniques, we examine the effects of a varying free surface on the distribution of buoyant particles. We simulate particle dispersal scenarios under a progressive deep water wave train as well as under an idealized wave spectrum. This work implies that waves on the surface of the ocean can affect the instantaneous distribution of particles. These distribution effects can provide insight into how to interpret surface observations of particles, such as microplastics, using net trawls.

3:52PM A02.00005 Detection of breaking waves in single wave gauge records of surface elevation fluctuations
- DAN LIBERZON, ALEXANDRU VREME, SAGI KNOBLER, Technion - Israel Institute of Technology — We report the development of a new method for accurate detection of breaking water waves, which addresses the need for an accurate and cost-effective method that is independent of human decisions. The new detection method, which enables the detection of breakers using only surface elevation fluctuation data, is capable of detecting breakers from a single wave gauge. Furthermore, we develop a new methodology for research relating to water waves and wind-wave interactions. According to the proposed method, detection is based on the use of the Phase-Time Method (PTM) to identify breaking-associated patterns in the instantaneous frequency variations of surface elevation fluctuations. A wavelet-based pattern recognition algorithm is devised to detect such patterns and provide accurate detection of breakers in the examined records. Validation and performance tests, conducted using both laboratory and open sea data, including mechanically generated and wind-forced waves, are reported as well. These tests allowed derivation of a set of parameters assuring high detection accuracy rates. The method is shown to be capable of achieving a positive detection rate exceeding 90 percent.

4:05PM A02.00006 Predicting the breaking onset and strength of gravity water waves in arbitrary depth
- Morteza Derakhhti, APL-UW, Jim Kirby, UD, Jim Thompson, APL-UW, Stephan Grilli, URI, Mike Banner, UNSW — We introduce a robust and local parameterization to predict the breaking onset and breaking strength of 2-D and 3-D gravity water waves in arbitrary depth. We use a LES/VOF model to simulate nonlinear wave evolution, breaking onset and post-breaking behavior for representative cases of 2-D and 3-D focused wave packets, modulated wave trains, regular and irregular waves propagating over various bed topographies featuring deep water, intermediate depths, and the shallow surf zone. We also use a 2D potential flow solver using BEM to simulate nonlinear wave evolution, focusing on breaking onset behavior. The new parameterization relates the breaking strength to a breaking strength predictor $\Gamma$ defined as the normalized rate of change of $B = U/C$ following the wave crest (with $U$ the water velocity at the crest and $C$ the crest celerity). We show that the breaking onset criterion proposed by Barthelemy et al. (2018) in deep water is also effective in shallow water (i.e., when $B > B_{th} \sim 0.85$ then breaking is imminent). The new parameterization is local and can handle multiple breaking events in space and time. The implementation of the new parameterization is convenient and efficient in phase-resolving models such as Boussinesq and HOS models.

4:21PM A02.00007 Numerical simulation of nonlinear and dispersive wave breaking in arbitrary depth
- Morteza Derakhhti, APL-UW, Jim Kirby, UD, Jim Thompson, APL-UW, Stephan Grilli, URI, Mike Banner, UNSW — We introduce a robust and local parameterization to predict the breaking onset and breaking strength of 2-D and 3-D gravity water waves in arbitrary depth. We use a LES/VOF model to simulate nonlinear wave evolution, breaking onset and post-breaking behavior for representative cases of 2-D and 3-D focused wave packets, modulated wave trains, regular and irregular waves propagating over various bed topographies featuring deep water, intermediate depths, and the shallow surf zone. We also use a 2D potential flow solver using BEM to simulate nonlinear wave evolution, focusing on breaking onset behavior. The new parameterization relates the breaking strength to a breaking strength predictor $\Gamma$ defined as the normalized rate of change of $B = U/C$ following the wave crest (with $U$ the water velocity at the crest and $C$ the crest celerity). We show that the breaking onset criterion proposed by Barthelemy et al. (2018) in deep water is also effective in shallow water (i.e., when $B > B_{th} \sim 0.85$ then breaking is imminent). The new parameterization is local and can handle multiple breaking events in space and time. The implementation of the new parameterization is convenient and efficient in phase-resolving models such as Boussinesq and HOS models.

4:39PM A02.00008 Validation and performance tests of a new technique for the detection of breaking waves
- Shahin Ranjarzadeh, Department of Mechatronic and Mechanical Systems University of Sao Paulo Brazil, Renato Picelli, Polytechnic School University of Sao Paulo Brazil, Raghavendra Sivapuram, Structural Engineering, University of California, San Diego, Emilio Carlos Nelli Silva, Department of Mechatronic and Mechanical Systems University of Sao Paulo Brazil. — This work presents a coupled topology optimization methodology to predict the breaking onset in a fluid flow channel. The methodology proposed here aims to apply the Topology Optimization of Binary Structure (TOBS) method to the development of new methodologies for research relating to water waves and wind-wave interactions. According to the proposed method, detection is based on the use of the Phase-Time Method (PTM) to identify breaking-associated patterns in the instantaneous frequency variations of surface elevation fluctuations. A wavelet-based pattern recognition algorithm is devised to detect such patterns and provide accurate detection of breakers in the examined records. Validation and performance tests, conducted using both laboratory and open sea data, including mechanically generated and wind-forced waves, are reported as well. These tests allowed derivation of a set of parameters assuring high detection accuracy rates. The method is shown to be capable of achieving a positive detection rate exceeding 90 percent.

4:55PM A02.00009 Inertial effects on wind-driven mixing in the coastal ocean
- Alexei G. Zheligov, Fred K. Schmugge, Broomfield, CO, United States — Inertial effects are important for coastal ocean processes, but they are not always easy to study and quantify. In this work, we present results of a field study of wind-driven mixing in the coastal ocean near Long Beach, California. We use a combination of direct measurements of wind stress and wave energy, and modeling to simulate the effects of wind and waves on the mixing process.

5:15PM A02.00010 A computational framework for simulating the transport of fine sediments in estuarine and coastal environments
- Mohammad Bagheri, Entebbe, Uganda, United States — A computational framework for simulating the transport of fine sediments in estuarine and coastal environments is presented. This framework is based on the coupling of a 3D hydrodynamic model with a 3D transport model. The hydrodynamic model is based on the Reynolds-averaged Navier-Stokes equations, while the transport model is based on the advection-diffusion equation.

5:35PM A02.00011 Toward a better understanding of sediment transport in estuarine and coastal environments
- Mohammad Bagheri, Entebbe, Uganda, United States — This talk presents a new approach for simulating sediment transport in estuarine and coastal environments. The approach is based on a coupled hydrodynamic-transport model that is capable of simulating the complex processes involved in sediment transport, including advection, diffusion, and deposition.

5:55PM A02.00012 An Eulerian method for mixed soft and rigid body interactions in fluids
- Xiaolin Wang, Harvard University, Ken Kamrin, Massachusetts Institute of Technology, Chris Rycroft, Harvard University — Fluid-structure interaction problems are encountered in many engineering and biological applications. In this work, we present an Eulerian approach for fluid-structure interactions that is simple to implement and capable of simulating multi-body interactions. When the solid is rigid, a projection step is formulated as a composite linear system that simultaneously enforces the rigidity and incompressibility constraints. When the solid is soft, a reference map technique is applied to characterize the body deformation in an Eulerian framework. Several examples including single soft and rigid flags, multiple rigid bodies, and soft-rigid combinations will be presented, with potential applications to biological systems.

6:15PM A02.00013 A soft material arena meter
- Johan Sundin, Shervin Bagheri, KTH Mechanics, Royal Institute of Technology, Katherina Kokmanian, Matthew Fu, Marcus Multmark, Mechanical and Aerospace Engineering Department, Princeton University — Micro air vehicles (MAVs) often operate in wind speeds of the same magnitude as their own velocity. In order to navigate and control them efficiently, there is a need to estimate the wind speed. Because of their low weight, around 100 g, a wind speed sensor must be small and lightweight. The sensor must also be able to withstand impulses from impacts or wind gusts. Most conventional sensor concepts today are unable to fulfill these requirements. We suggest a new flow sensor concept, based on electrically conductive soft materials, denoted organic elastic filament velocimetry (OEFV). This technique estimates flow velocity by measuring the strain a polymer ribbon in the flow experiences. The polymer ribbon can withstand strains of the order of $\epsilon \sim 1$, making it extremely durable, in contrast to most conventional materials. The ribbon is manufactured in polydimethylsiloxane (PDMS) and made piezoresistive by adding a thin layer of silver nanowires, so that the resistance of the ribbon can be related to the flow velocity. The large aspect ratio of the ribbon (length to thickness) simplifies the description of the flow around it and amplifies the sensor output. A fairly simple model of the sensor behavior is constructed and compared to experimental data.

---

1Supported by SSF (FFL15:00001) and by AFOSR (FA9550-16-1-0170)
A feasibility study on the potential for employing piezoelectric cantilever beams as vortex flow sensors. AMIR DANESH-YAZDI, Rose-Hulman Institute of Technology, OLEG GOUSSHCHA, Manhattan College, YIANNIS ANDREOPoulos, City College of New York — The Train of Frozen Boxcars (TFB) model was previously developed to study the effective one-way coupling of the force due to the advection of a vortex or train of vortices over a stiff piezoelectric cantilever beam. The TFB approach involves the advection of several boxcars of different amplitudes, widths and separations as a model for the fluidic force acting on the beam. In this talk, we explore utilizing the TFB model as a way to deduce the force that acts on the piezoelectric beam due to the vortex. This study is aimed to predict vortex properties associated with vortex flow such as vortex circulation, diameter and separation between vortices. Preliminary results indicate that while the original TFB model can predict the properties of an individual vortex rather well, the model requires tweaks in order to better predict vortex properties when a train of vortices are involved.

Pendulum in a Flow: Case of a Balanced Pendulum. ARIANE GAYOUT, Laboratoire de Physique, ENS de Lyon, CNRS, France, ARMANn GYLFASON, School of Science and Engineering, Reykjavik University, Iceland, NICOLAS PLIHON, MICKAEL BOURGOIN, Laboratoire de Physique, ENS de Lyon, CNRS, France — Fluid-structure interactions are the basis of the complexity of Aerodynamics, enhancing resonance in structures and turbulence in flows. Even simple systems like a pendulum can become more complex, as a hysteretic bistability shows up for a range of flow velocities when the pendulum confronts a flow. This is predicted by a simple balance of weight and aerodynamical forces, but non stationary response can be seen through spontaneous transitions between both stable positions. This dynamic can also be observed when the weight of the pendulum. By analyzing trajectories in different phase spaces, we recover a stochastic measurement of the drag and lift coefficients. Moreover, the pendulum oscillates around the horizontal at a frequency that is linked to the evolution of the normal drag coefficient with the angular position of the pendulum. The instantaneous lift and drag coefficients inferred from the dynamical behavior of the pendulum seems to be governed by the dynamical vortex shedding phenomena, which we currently investigate experimentally.

Segregation modeling of binary suspensions. STANY GALLIER, MATHIEU PLAUD, ArianeGroup — This study proposes a continuum model able to describe particle segregation in bidisperse non-colloidal suspensions. It is based on the Morris-Boulay suspension balance model [Morris and Boulay, J. Rheol., 1999] which has been here extended to two classes of particles with different sizes. Doing so gives rise to new quantities (basically, normal or shear viscosity by particle class) that are not available experimentally. Indeed, only the overall suspension normal and shear viscosity are reported and it is not known how they split with respect to particle size class. We therefore consider particle-resolved Stokes simulations of binary suspensions to address this point. Parametric simulations are conducted for different size ratio, total volume fraction and large/small fraction ratio. We show that both shear and normal viscosities are approximately proportional to volume fraction ratio. This allows to provide a simple closure for the model. It is then incorporated in a flow solver and successfully compared to available experiments of segregation in channel flows.

Simulating cross stream migration of hard spheres in a dilute suspension. NILANKA I. K. EKANAYAKE, JOSEPH D. BERRY, ANTHONY D. STICKLAND, DAVID E. DUNSTAN, The University of Melbourne, STEVEN K. DOWER, INEKE L. MUIR, CSL Limited, DALTON J. E. HARVIE, The University of Melbourne — In flowing suspensions, hydrodynamic forces can move particles across the streamlines and cluster at certain radial positions within a pipe. In the biological flow context, this cross-stream migration is widely used in cell sorting microfluidic applications to separate diseased cells based on size and other physical characteristics. This study employs a multi-fluid model to predict the solid concentration profiles of a mono-disperse suspension of hard spheres flowing at low particle Reynolds numbers. The lateral migration is modelled using wall-shear and shear rate gradient hydrodynamic lift forces and inter-particle hydrodynamic collision forces. Brownian and shear induced diffusion forces are modelled as functions of solid concentration gradients and shear rate gradients. The effect of bulk concentration on solid distribution is examined and compared against experimental data. At highly dilute conditions particles accumulate at 0.6 radius away from the centerline exhibiting the “tabular pinch” effect caused by lift forces. With increasing bulk concentration, particles gradually move towards the centerline due to diffusion forces. This study demonstrates that the bulk concentration has a significant impact in determining solid distribution profiles.
3:26PM A04.00003 Lubrication Corrections for Brownian Suspensions Above a Wall1
   BRENNAN SPRINKLE, ALEKSANDAR DONEV, YIXIANG LUO, Courant Institute of Mathematical Sciences — Lubrication effects play an
   important role in the dynamics of dense suspensions of passive or active microscale particles. In order to quantitatively measure bulk properties
   in these types of suspensions, the near-field lubrication flows must be accurately resolved. We present an efficient numerical method to simulate
dense Brownian suspensions of many particles above a bottom wall. Our method includes lubrication effects while maintaining a computational
complexity which scales linearly in the number of particles. Examples to demonstrate the effectiveness of our method include: the sedimentation
of particles over an inclined plane, as well as collective motion in a suspension of magnetic ‘rollers’.
1DMR-1420073, RTG/DMS-1646339

3:39PM A04.00004 Modeling and Stokesian-Dynamics Simulations of Friction in Colloidal Suspensions1, GERALD WANG, MIT Chemical Engineering; CMU Civil and Environmental Engineering, JAMES SWAN, MIT Chemical Engineering — As the gap between particles in a suspension approaches zero, inter-particle frictional forces can become
non-negligible. These forces may strongly influence structure and transport in systems with a significant number of particle-particle contacts,
as such as sticky colloidal suspensions. Understanding the influence of microscopic frictional contacts on macroscopic suspension properties is vital
for modeling many rheological phenomena, including discontinuous shear thickening. In this talk, we present computational and theoretical work
exploring the role of friction in colloidal suspensions, in which friction is treated as a hydrodynamic resistance to sliding between particles near
contact. Our Stokesian-Dynamics simulations demonstrate that the frictional suppression of both the translational and rotational diffusivities
can be related to the friction coefficient using a simple power-law scaling. To explain this result, we develop a spectral model that quantifies
the impact of friction on the hydrodynamic resistance tensor. We then show that a mathematical model of the suspension as a graph, which
tracks connections between neighboring particles, can be used to explain the observed dynamics. We also discuss experimental results.
1This work has been supported partially by National Science Foundation award #1854376 and partially by Army Research Office award
#W911NF-18-1-0356.

3:52PM A04.00005 Particle migration of colloidal and Brownian suspensions in both Poiseuille and circular Couette flow1, CHANGWOO KANG, PARISA MIRBOD, University of Illinois at Chicago — The flow of neutrally buoyant and hard-sphere colloidal particles concentrated in a Newtonian viscous fluid is examined by direct numerical
simulations (DNS) at various bulk particle volume fraction (0.1 ≤ φp ≤ 0.5) and Peclet number (10−2 ≤ Pe≤103). We use the diffusive flux
model (DFM) to describe the behavior of suspensions and employ the viscosity introduced by de Kruijf et al. [J. Chem. Phys. 1985] which
is given as a function of shear rate and volume fraction. First, we consider pressure-driven flow of colloidal particles in a channel. For low
Pe number the concentration profile flattens, as Pe grows the influence of Brownian motion diminishes and the distribution of concentration
reaches the profile of non-colloidal suspensions flow. Also, as Brownian motion becomes dominant, the volume flow rate decreases steadily. We
then study a circular Couette flow of colloidal suspensions where the inner cylinder rotates with a constant angular velocity and the outer one is
fixed. The concentration profile flattens out and the local shear rate decays with the reduction of Pe number. The torque acting on the outer
inlet cylinder builds up due to colloidal suspensions.
1This work has been supported partially by National Science Foundation award #1854376 and partially by Army Research Office award
#W911NF-18-1-0356.

4:05PM A04.00006 Graphene nanoplatelets attain a stable orientation in a shear flow: an investigation on the role of Brownian fluctuations1, CATHERINE KAMAL, SIMON GRAVELLE, LORENZO BOTTO, Queen Mary University of London — We study theoretically the rotational dynamics of a rigid graphene nanoplatelet
suspended in a simple shear flow. We have recently shown that in the infinite Peclet number limit a rigid platelet presenting the interfacial
hydrodynamic slip properties of graphene does not follow the periodic rotations predicted by Jefferys theory, but rather aligns itself at a small
inclination angle α with respect to the flow. This unexpected result is due to the low tangential friction at the graphene-solvent surface (the slip
lengths for many 2D materials/solvent combinations can amount to several tens of nanometers). By analyzing the Fokker-Plank equation for
the orientation distribution function for decreasing Peclet numbers, we show that the platelet fluctuates about α until a slip length dependent
critical Peclet number is reached. Below this value, Brownian forces are large enough to produce full rotations, bringing the particle outside of
a hydrodynamic potential well. In the stable region, the effective viscosity of a dilute suspension of graphene platelets is predicted to drop by
at least a factor of 2 for typical slip length values.
1ERC project: FLEXNANOFLOW

4:18PM A04.00007 Shear-induced exfoliation of graphite nanoplatelets into graphene: insights from non-equilibrium molecular dynamics1, SIMON GRAVELLE2, CATHERINE KAMAL, LORENZO BOTTO3, School of Engineering and Materials Science, Queen Mary University of London, E1 4NS, London, UK — Graphite nanoplatelets suspended in a shear flow can be exfoliated into single-layer or few-layers graphene. Such liquid-phase exfoliation
method is a promising technique for the production of graphene on large scales, but the micro and nanoscale fluid-structure processes controlling
exfoliation are not understood. Here we use molecular dynamics simulations of a defect-free graphite nanoplatelet suspended in a shear flow to
caracterise the exfoliation dynamics and measure the critical shear rate above which exfoliation occurs, comparing the effect of using different
solvents (water and NMP). The measured critical shear rates are compared with a simple theoretical model due to Chen et al. (Chem. Commun.,
48, (2012) 3703-3705) based on a balance between the work done by viscous shearing forces and the change in interfacial energies upon layer
sliding. We found it difficult to reconcile the molecular dynamics results with the model. The results obtained so far highlight the importance of
hydrodynamic slip at the liquid-graphene interface, and the effect of graphene edges on hydrodynamics and interfacial mechanics. Both effects
are incompletely included in the model.
1ERC project FLEXNANOFLOW (n. 715475)
2Membership Pending
3other affiliation: Faculty of Mechanical, Maritime and Materials Engineering, Department of Process and Energy, TU Delft, the Netherlands

Saturday, November 23, 2019 3:00PM - 4:31PM –
Session A05 Compressible Flows: Shock Interactions and Shock Focusing 204 - Veronica
Eliassson, University of California San Diego
3:00PM A05.00001 Normal shock wave attenuation during propagation in straight and branching ducts with grooves. SEYED MEHDI MORTAZAWY, Embry Riddle Aeronautical University, KOSTAS KONTIS, School of Engineering, University of Glasgow, JOHN EKATERINARIS, Embry Riddle Aeronautical University — Experimental investigations and numerical simulations of normal shock waves of different strengths propagating inside ducts with roughness are presented. The roughness is added in the form of grooves. Straight and branching ducts are considered in order to better explore the mechanisms causing attenuation of the shock and the physics behind the evolution of the complex wave patterns resulting from diffraction and reflection of the primary moving shock. A well-established finite volume numerical method is used and further validated for several test cases relevant to this study. The computed results are compared with experimental measurements in ducts with grooves. Good agreement between high-resolution simulations and the experiment is obtained for the shock speeds and complex wave patterns created by the grooves. High frequency response time histories of pressure at various locations were recorded in the experiments. The recorded pressure histories and shock strengths were found in fair agreement with the two-dimensional simulation results as long as the shock stays in the duct. Overall, the physics of the interactions of the moving shock, and the diffracted and reflected waves with the grooves are adequately captured in the high-resolution simulations.

3:13PM A05.00002 Combating shock waves: A study of shock wave cancellation mechanisms within linear turbine blade cascades. JORGE NUNEZ, VERONICA ELIASSON, UCSD — Prior research has established that unsteady shock wave interactions on turbine blades negatively impact the fatigue life and performance of gas turbines. Current technology in gas turbines mitigates negative effects on turbine blades by way of cooling mechanisms. However, recent inventions seek to eliminate the adverse influence that unsteady shock wave interactions have on turbine blades by cancelling incident shock waves in gas turbines. The goal of this study is to ascertain the efficacy of shock cancellation mechanisms in gas turbines at preventing incident shock waves from reflecting off the turbine blades. A study of shock wave cancellation in linear turbine blade cascades has been performed to analyze the performance of shock wave cancelling mechanisms. CFD simulations are compared to shock tube experiments featuring high-speed schlieren photography and pressure measurements of unsteady shock wave impacts on linear turbine blade cascades. Different shock wave cancelling designs has been investigated and the benefits and drawbacks of the chosen designs will be discussed.

1This investigation is sponsored by the University of California San Diego NEW Scholars Undergraduate Research Scholarships (URS) program from the Academic Enrichment Programs (AEP) office and their support is gratefully acknowledged.

3:26PM A05.00003 Scaling of compressible turbulent mixing. EUNHYE AN, ERIC JOHNSEN, University of Michigan — The decay of homogeneous, isotropic turbulence is well understood based on Kolmogorov theory in the incompressible limit. However, the roles of compressibility and inhomogeneities on the turbulence phenomenology are less well known. For this purpose, we conduct direct numerical simulation (DNS) to investigate the behavior of turbulent mixing, in which homogeneous isotropic turbulence regions of different intensities are juxtaposed. We investigate the scaling of the decay of turbulent kinetic energy (TKE) for this compressible, inhomogeneous flow. By considering the turbulent energy balance equation, we determine scaling coefficients predicting the observed behavior by accounting for dilatation. This scaling is verified by the DNS results of turbulent mixing as well as turbulent/non-turbulent interfaces.

1This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number ACI-1548562.

3:39PM A05.00004 Direct Simulation of Fluid-Structure Interaction in a Hypersonic Compression Ramp Flow. BRYSON SULLIVAN, University of Illinois at Urbana-Champaign, THOMAS WHALEN, STUART LAURRENCE, University of Maryland, DANIEL BODONY, University of Illinois at Urbana-Champaign — Sustained hypersonic flight presents an enduring challenge to aircraft design and control. An extreme aerothermal environment acting on thin, multi-functional structures can yield significant fluid-thermal-structural interaction (FTSI) in control surfaces and/or a complete vehicle. While computationally efficient, the accuracy of reduced-order aerodynamic models can be compromised by local regions of subsonic/separated flow, highlighting the need for high-fidelity numerical simulations. The present talk outlines recent time-accurate FTSI simulations of a viscous flow at $M_{∞} = 6.0$ over a 35° compression ramp with an embedded compliant panel. Reduced-order models are compared directly to FTSI simulation data, and a simple modification is proposed which can improve the accuracy of shock-expansion/local piston theory predictions. A reduction in surface heat flux is observed for most compliant cases, while traditional heat transfer analogies were found to be reliable only for the rigid case. Oscillation of the reattachment point was found to synchronize with the motion of the compliant panel for large-amplitude vibrations.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0035.

3:52PM A05.00005 Simulation and Global Stability Analysis of a 35° compression ramp in a uniform Mach 6 flow. FABIAN DETTENRIEDER, DANIEL BODONY, University of Illinois at Urbana-Champaign — Aerothermal and structural loads of flows relevant to hypersonic vehicles pose a multi-disciplinary design problem. Of particular interest is the fluid-structure interaction associated with a deflected control surface at sufficiently high Mach number such that the compression shock sits on the control surface downstream of the corner, leading to large localized pressure and thermal loads. As a baseline for subsequent compliant analyses, we investigate the Mach 6 flow past a rigid 35° degree compression ramp attached to a finite-length flat plate, which together model the control surface geometry, by means of direct numerical simulation (DNS). The three-dimensional DNS results include the boundary layer development from the plate’s leading edge, through transition, and ultimate transition to turbulence. The unsteadiness of the separation bubble and observed ejection of mass and shocklets from the separation bubble are quantified and compared to experimental data. We use global stability analyses of the time-averaged flow to discuss the observed unsteadiness in terms of global modes.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0035.
The DSMC method, used to simulate hydrogen-air combustion. This method uses simulation particles that

Shriya Trivedi, R. Stewart Cant, John K. Harvey, University of Cambridge — A novel molecular simulation technique, namely

The authors thank AFRL, award: FA8651-17-1-004, and NSF CBET, award: CBET-1803592, for funding.

The novelty of this system lies in (1) the ability to produce shock waves consistent with blast type flow conditions; (2) its modularity by virtue of the driver; and (3) its application in gaseous environments. The outcome has been an experimental setup with novel application in air, proven repeatability, as well as initial qualitative and quantitative results in its two-dimensional, cylindrical format.

1The authors thank AFRL, award: FA8651-17-1-004, and NSF CBET, award: CBET-1803592, for funding.

Saturday, November 23, 2019 3:00PM - 4:31PM —
Session A06 Novel Techniques for Combustion Modeling 205 - Eric Climent, Institut de
Mecanique des Fluides de Toulouse

3:00PM A06.00001 Simulation of combustion processes using the DSMC method1.
Shriya Trivedi, R. Stewart Cant, John K. Harvey, University of Cambridge — A novel molecular simulation technique, namely

1This work has been performed using resources provided by the "Cambridge Service for Data Driven Discovery" (CSD3, http://csg3.cam.ac.uk)

3:13PM A06.00002 Reduced order description of diffusion flames with inertial manifold
theory1, Maryam Akram, Venkat Ramani, University of Michigan, Ann Arbor — Resolving all dynamical scales in reacting

3:26PM A06.00003 Numerical and Experimental Study of Piloted Liquid Spray
Flames, Dorrin Jarrahbashi, Yejun Wang, Salar Taghizadeh, Waruna Kulatilaka, Texas A&M University, Texas A&M UNIVERSITY TEAM — Liquid-fuel spray flames are the primary mode of energy conversion in many high-power-density practical combustion devices. A modified, flat-flame McKenna burner fitted with a direct-injection high-efficiency nebulizer is used to produce piloted liquid-methanol spray flames. A 3D computational model is developed which comprised of compressible continuous gas phase using URANS in conjunction with two-way coupled Eulerian-Lagrangian spray modeling approach along with a partially-stirred reactor combustion model. Model predictions of Hydroxyl (OH) and carbon monoxide (CO) in the radial and axial directions at different flame locations are compared with laser-based imaging measurements. OH profiles are obtained using nanosecond planar laser-induced fluorescence (PLIF) for characterizing the reaction zone and temperature, while 2D images of CO are obtained via two-photon laser-induced fluorescence (TPLIF) using femtosecond-duration laser pulses. The computational model predicts the general trends of OH, temperature and CO profiles well at certain heights above the burner surface. The sensitivity of the model to the droplet size distribution, pilot flame temperature, and the co-flow temperature are discussed.
3:39PM A06.00004 A novel deep learning framework for efficient parameterization of high-dimensional flamelet manifolds. OPEOLUWA OWOYELE, PRITHWISH KUNDU, PINAKI PAL, Argonne National Laboratory — Tabulated flamelet models are limited by the curse of dimensionality, wherein the computational memory required to store multidimensional flamelet lookup tables grows exponentially as the number of independent variables increases. In addition, it requires a vast increase in code complexity to perform interpolations in higher dimensions. One promising technique is to train an ANN \textit{apriori} based on the flamelet table, and use the trained model during run-time to obtain the species mass fractions as functions of the independent variables. Results from a previous study showed that training accuracy and inference speed can be improved if the multidimensional data set is bifurcated and separate ANNs are used for different regions. In this work, an \textit{apriori} study is performed using a physically meaningful divide-and-conquer approach, where an ensemble of deep neural networks is trained to represent the relevant physics in different regions of the flamelet manifold, as supervised by a gating network. The method is applied to 4-dimensional and 5-dimensional flamelet tables representing spray combustion in a constant volume chamber (ECN spray A) and a compression-ignition engine, respectively. The technique is demonstrated to result in a further decrease in the computational cost for a given accuracy. Moreover, the physical meaning of the obtained partitions is also discussed.

3:52PM A06.00005 Mass transfer coupled to chemical reaction through a random array of fixed porous particles. ERIC CLIMENT, Institut de Mecanique des Fluides de Toulouse, MOSTAFA SULAIMAN, ABDELKADER HAMMOUTI, IFP-EN, ANTHONY WACHS, University of British Columbia, UNIV. OF BRITISH COLUMBIA COLLABORATION — We have studied by means of numerical simulations the effect of a first order irreversible chemical reaction on mass transfer for two-phase flow systems in which the continuous phase is a fluid and the dispersed phase consists in catalyst spherical particles. The reactive solute is transported by the fluid flow and penetrates through the particle surface by diffusion. The chemical reaction takes place within the bulk of the particle. We handle the problem by coupling mass balance equations for internal-external transfers through the particle surface. We propose a model to predict mass transfer coefficient accounting for the external convection-diffusion along with internal diffusion-reaction. For the simulation of multi-particle systems, we have implemented a Sharp Interface Method to handle strong concentration gradients in DLM/FD framework. We validated the method thoroughly thanks to comparison with analytical solutions in case of diffusion, diffusion-reaction and by comparison with previously established correlations for convection-diffusion mass transfer. Finally, we study the configuration of a fixed bed of catalyst particles. We introduce a model that accounts for the solid volume fraction, in addition to the aforementioned effects and compare to numerical simulations.

4:05PM A06.00006 Quasi-static Eulerian framework for thermomechanical heterogeneous solid propellant simulations. TADBHAGYA KUMAR, THOMAS JACKSON, University of Florida — Heterogeneous solid propellants find prevalent use in space missions and thus understanding physics that influences their performance becomes critical to design. Numerical simulation of propellant combustion is complex owing to multiphysics nature with a burning surface that propagates through pyrolysis law. Of interest is to predict the thermomechanical stresses and strains that might lead to damage/failure of the constituent materials of the propellant under combustion. In this work, a Eulerian framework for thermomechanical simulations of propellant combustion using the hypoelastic equation is presented. A quasi-static projection method is motivated through scaling analysis and a finite element based weak form is employed to deal with material moduli differences. The simulations are carried out in two-dimensional propellant packs and results are presented for the temperature, stress, and their effect on propellant burn rate.

4:18PM A06.00007 A Novel Experiment-Based Framework for Turbulent Combustion Modeling. RISHIKESH RANADE, TAREK ECHEKKI, North Carolina State University — A novel framework for turbulent combustion modeling is presented. The framework is based on the construction of conditional means and joint scalar PDFs from multiscalar measurements in flames based on the parameterization of composition space using principal component analysis (PCA). The resulting principal components (PCs) act as both conditioning and transported variables. Their chemical source terms are constructed starting from instantaneous temperature and species measurements using a variant of the pairwise mixing stirred reactor (PMSR) approach. A multi-dimensional kernel density estimation (KDE) approach is used to construct the joint PDFs in PC space. The PDFs’ dimension corresponds to the number of retained PC’s and the composition conditional means provide measures of the unconditional means for the closure terms: the mean PCs chemical source terms and the density. The framework is demonstrated a priori and posteriori using data from different flames.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A07 Aerodynamics: Airfoil

3:00PM A07.00001 Data-driven analysis of vortex dynamics around a sinusoaidally pitching airfoil. KARTHIK MENON, RAJAT MITTAL, Johns Hopkins University — Data-driven methods to analyze fluid flows have been recently gaining popularity in many subdomains of fluid dynamics. This has been primarily driven by our improved ability to generate large, high-quality data sets, and efforts to extract patterns from large amounts of data in an efficient manner. This talk will describe our work to understand the dynamics of aeroelastic flutter from one such data set consisting of over 500 simulations of a sinusoaidally pitching airfoil under different conditions. In particular, the focus will be on the analysis of the vortex dynamics close to the surface of the airfoil as well as in the wake, which is instrumental in driving flutter. We describe a new dynamic mode decomposition (DMD) formulation that allows us to decompose the flow in the vicinity of the moving airfoil and also discuss data-driven clustering methods to identify distinct vortex patterns in this complex flow.

3:13PM A07.00002 Fluttering, twisting and orbital motions of wall-mounted flexible plates. YAQING JIN, The University of Texas at Dallas, SHYUAN CHENG, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — The dynamics of wall-mounted flexible plates under inclined flows was studied for a variety of Cauchy numbers using theoretical arguments and laboratory experiments. Particle tracking velocimetry and a high-resolution force sensor were used to characterize the plate dynamics and aerodynamic force. Results show three distinctive modes of tip oscillations, which are modulated by the structure dynamics and flow. The first mode is characterized by small-amplitude motions occurring by small Cauchy number. Past this condition, the motions are dominated by unsteady twisting patterns. The onset of this mode is characterized by a sharp increase of the force fluctuation intensity. At sufficiently high Cauchy number and flow inclination, the plate may undergo a third mode dominated by large-scale tip orbits about the mean bending. We propose a formulation to estimate the critical Cauchy number as a function of inclination angle, which agrees well with experiments.
3:26PM A07.00003 On the multiscale oscillations of a hinged plate under stratified turbulence1, SHYUAN CHENG, Department of Mechanical Science and Engineering, University of Illinois, Urbana, IL, 61801, YAQING JIN, Department of Mechanical Engineering, The University of Texas, Richardson, TX, 75080, LEONARDO P. CHAMORRO, Department of Mechanical Science and Engineering, University of Illinois, Urbana, IL, 61801 — Wind-tunnel experiments were conducted to quantify the unsteady motions of a rigid plate under stratified turbulence containing spatially-varied, energetic vortices. The structure was able to oscillate around a vertical axis located at a quarter chord length from the leading edge. Vertically-oriented, von-Karman vortices that varied in frequency and strength along the vertical span of the plate were imposed using a variety of tapered cylinders. Telemetry and hotwire anemometry were used to characterize the motions of the plate, and wake at selected locations. Results show that the plate oscillation is dominated by two distinct modes. One of them, f_p, corresponds to the mean flow-induced oscillation frequency; whereas the other, f_v, is related to the vortex shedding cells from the non-uniform cylinders. These frequencies exhibited distinct trends with the distance from the cylinders, and the distribution of the coherent motions, which were determined by the tapered ratio of the cylinders. A simple model is proposed to estimate f_v.

3:39PM A07.00004 Flutter-Enhanced Mixing: Flow-Induced Flutter of Flexible Membranes in Small Scale Mixers2, AARON RIPS, NASA/Jacobs Space Exploration Group, RAJAT MITTAL, Johns Hopkins University — Flow-induced flutter of flexible membranes in small scale mixers is explored. Fully-coupled fluid-structure-scale interaction (FSI) simulations are used to examine the mixing enhancement in duct-style small scale mixers due to fluttering flags and the sensitivity of the dynamics and the mixing performance to Reynolds number is examined. The fluid and structural dynamics are explored and the resultant mixing enhancement is characterized according to existing performance measures as well as two new analysis techniques. A new performance measure called the Equivalent Mixing Length is developed to examine the connection between increased mixing performance and the corresponding increase in pressure loss which can serve as a measure of performance efficiency of flutter mixers or of any micromixer design. Additionally, a technique is developed to extract an estimate of the interface path length in the flow so as to characterize the impact of increased convection due to the presence of the flutter mixer. Through these measures it is shown that flutter mixers can significantly improve the mixing performance of a small scale mixer over a very small duct length by way of generating vortical structures even at low Reynolds numbers which leads to complex stretching and folding of the interface in the mixer.

3:52PM A07.00005 Nonlinear Stability Characteristics of an Elastically Mounted Pitching Wing1, YUANHANG ZHU, YUNXING SU, KENNETH BREUER, Brown University — We study the nonlinear stability boundaries of an elastically mounted pitching wing in a water flume, with the wing structural dynamics (inertia I, stiffness k, damping b) simulated using a cyber-physical system. We fix b to be small and systematically vary k at different I to test for the onset and extinction of self-excited oscillations. We find that as k increases, the system bifurcates from a fixed point towards small-amplitude limit cycle oscillations via a subcritical bifurcation, which features hysteretic bistability and an abrupt amplitude jump at the bifurcation. At this I, the wing pitching frequency f_p locks onto its structural frequency f_s, indicating dominating structural force. Force and PIV measurements reveal the emergence of a secondary leading edge vortex (LEV) after the shedding of the primary LEV. As I decreases, the width of the bistability region shrinks. When I is sufficiently low, the pitching amplitude changes gradually with k without hysteresis, revealing a supercritical bifurcation. At this I, f_s is relatively constant and lower than f_p, indicating dominating fluid force. The secondary LEV is not present. We also report the effect of sweep angles on the stability boundaries.

4:05PM A07.00006 Periodic Loading of a Mach 4 Boundary Layer over a Compliant Surface by an Oscillating Shock Generator1, MALLORY NEET2, JOANNA AUSTIN3, California Institute of Technology — A significant challenge in designing hypersonic vehicles is developing predictive models for aerodynamic loads associated with turbulent boundary layers and shock boundary layer interactions on compliant surfaces. Shock-boundary layer interactions on high-speed vehicles can lead to pressure fluctuations which can couple to structural modes and lead to premature high-cycle fatigue and ultimately component failure. In collaboration with simulations, some experimental studies can give insight into the degree of fluid-structure coupling under prescribed loading conditions. In this presentation, the flow response over a 0.2 mm thick compliant steel panel under dynamic loading from an oscillating shock generator is investigated in the Caltech Mach 4 Ludwieg tube. Euler simulations were performed to design the geometry and location of the shock generator, predict the amplitude of the oscillation required to drive the pressure wave across the compliant surface, and to calculate the pressure rise on the compliant surface. The flow response is characterized using fast-acting pressure transducers, high-speed schlieren images, and porous fast-response pressure sensitive paint and the panel response using laser doppler vibrometry. 

4:18PM A07.00007 Anemometry from visual observations of fluid structure interactions1, JENNIFER L. CARDONA, MICHAEL F. HOWLAND, JOHN O. DABIRI, Stanford University — Visual observations of objects interacting with the wind contain information about local wind conditions such as wind speed. These visual encodings can potentially be leveraged to measure wind speeds using videos of pre-existing objects in an environment (e.g. flapping flags or swaying trees). We propose a data driven approach that leverages deep learning methods to predict wind speeds given video recordings of fluid structure interactions. Video clips of flags and trees moving due to naturally occurring wind are used to estimate wind speeds through the application of a convolutional neural network followed by a recurrent neural network. Physical parameters of the observed objects are used to aid in understanding limitations of model performance.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A08 Surface Tension I 212 - Hassan Masoud, Michigan Tech

3:00PM A08.00001 Rifts in Rafts1, KHA-I TO, DANIEL HXNER, VINCENZO VITELLI, SIDNEY NAGEL, University of Chicago — Two-dimensional particle rafts are single-layers of aggregated sub-millimeter polydisperse particles floating at an air-fluid interface. The material failure of such rafts under an applied extensional load has a morphology that appears to be distinct from other known fracture modes. At higher extensional shear rates, numerous small-scale cracks are distributed diffusively throughout the entire system; at low strain rates, the distance between adjacent cracks increases. The characteristics of this distributed failure also depend on the surface tension and viscosity of the underlying fluid. To decrease the influence of secondary flows, we perform experiments by changing the liquid level in the tanks with inclined walls so that we are able to increase the area accessible to the rafts as the liquid height changes. This results in an expansion in quasi-1D (with and without boundaries) and isotropic 2-D expansion in the linear and cylindrical geometries respectively. We simulate this behavior with a model based on weak interparticle forces coupled to an expanding underlying metric.
3:13PM A08.00002 Morphology Regulation of Liquid-Gas Interface on Bioinspired Super-Repellent Surface. YAOLEI XIANG, PENCYU LV, HUILING DUAN, Department of Mechanics and Engineering Science, College of Engineering, Peking University, Beijing 100871, P. R. China — Bioinspired underwater super-repellent surfaces have many excellent properties, which attribute to the air mattress trapped on the surface. However, instability and collapse of the underwater slippery air mattress hinder its applications, after which the air mattress is difficult to recover. Here, we find that the unique hierarchical structures on the leaf surface of a famous invasive floating fern, Salvinia, have the capacity to replace the impregnated water with air and entirely recover the air mattress. We reveal the underlying mechanisms of the recovery process. The interconnected wedge-shaped grooves on the bottom are key to the recovery, which spontaneously transport the replenished air to the entire surface governed by a gas wicking effect. Moreover, inspired by the nature of Salvinia, we fabricated artificial Salvinia surfaces using three-dimensional printing technology, which successfully achieve a complete recovery of a continuous air mattress to exactly imitate the super-inflatable capability of Salvinia leaves.

3:26PM A08.00003 Collapse of a Compressed Granular Raft1. BEN DRUECKE, XIANG CHENG, SUNGYON LEE, University of Minnesota — Rigid, passive particles at the interface between two fluids provide a compression-resistant interface, thus opposing area-minimizing interfacial energy and providing the stabilizing effect characteristic of Pickering emulsions. We experimentally and analytically investigate the behavior of a flat particle raft under isotropic compression. A granular raft of glass spheres with diameter in the range of 0.1 to 2 mm is formed on a fluid-fluid interface within a conical funnel. Axissymmetric compression of the raft is imposed by draining fluid from the funnel. Two distinct modes of raft deformation and collapse are observed, depending on particle size and the two fluids forming the interface. In the first mode, individual particles fall from the raft in seemingly uncorrelated events. In the second, the raft collectively deforms and creases, akin to a Rayleigh-Taylor instability. We analytically and experimentally examine the two modes of raft collapse as a function of particle size, fluid densities and surface energies.

1 Funded by the National Science Foundation through the University of Minnesota MRSEC under Award Number DMR-1420013

3:39PM A08.00004 Deformation of quasi-two-dimensional drops traveling in a microchannel.1. PABLO MARDONES MUÑOZ, MARÍA LUISA CORDERO GARAYAR, Universidad de Chile — When water and oil are injected into a shallow channel, water droplets form in a quasi-two-dimensional geometry. In an equilibrium situation the water-oil interface of the drops adopts a circumferential shape of radius R to minimize the energy. However, when a pressure gradient puts the system out of equilibrium, the shape of the droplets is modified to respond to the strains on their edge.

To characterize the deformation of the drops, we have measured the interface shape and decomposed it into Fourier modes. We focus on the evolution of the drop deformation, since their formation until they reach their equilibrium shape. By varying the experimental conditions, we study drop shape and its evolution as a function of the capillary, Ca, number and the confinement of the drops in the shallow channel, d. We have found a scale law in which the steady-state root mean square deformation of the drops is proportional to the confinement d to the fourth power.

1Ph.D. student finance by CONICYT National PhD-fellowship: Beca Nacional (2018) No. 21181182 and Fondecyt project No. 1170411

3:52PM A08.00005 ABSTRACT WITHDRAWN –

4:05PM A08.00006 Forward, halted, and reverse motion of an active particle atop a finite liquid layer1. SAEED JAFARI KANG, Michigan Tech, JONATHAN ROTHSTEIN, University of Massachusetts Amherst, HASSAN MASOUD, Michigan Tech — We examine the mobility of a chemically active particle straddling the interface between a liquid layer of finite depth and a semi-infinite layer of gas. A surface-active agent is released asymmetrically from the particle that locally lowers the liquid surface tension. It is commonly presumed that the uneven distribution of surface tension and the associated Marangoni flow lead to the propulsion of the active particle opposite to the release direction, where the surface tension is higher. This is considered forward motion. However, our recent theoretical analysis (in the limits of negligible inertia and diffusion-dominated transport of the active agent) has shown that this trend may be reversed for certain particle shapes and shallow enough liquid layers. Advancing beyond the Stokes regime, here, we numerically study the Marangoni-driven motion of oblate spheroidal particles for a wide range of release rates and subject to various degrees of confinement, represented by the thickness of the liquid film. We show that this trend may be reversed for certain particle shapes and shallow enough liquid layers. Advancing beyond the Stokes regime, here, we numerically study the Marangoni-driven motion of oblate spheroidal particles for a wide range of release rates and subject to various degrees of confinement, represented by the thickness of the liquid film. We show that this trend may be reversed for certain particle shapes and shallow enough liquid layers.

1Financial support from the National Science Foundation under Grant Nos. CBET-1749634 (H.M.) and CBET-1705519 (J.P.R) is acknowledged.

4:18PM A08.00007 Flow through a Catenoid: The Fluid Tube1. MACKENZIE DUCE, California Polytechnic State University at San Luis Obispo, AARON BROWN, DANIEL HARRIS, Brown University — Minimal surfaces have been studied for centuries by mathematicians, and can be readily realized using fluid films such as a film of soapy water. In particular, two rings are known to be connected by a catenoid-shaped film, however, beyond a critical ring spacing the catenoid solution fails to exist and the structure collapses. In this work, we experimentally investigate a variation of this classic problem by introducing steady flow through the catenoid structure formed by an oil film in water. We demonstrate that the flow robustly stabilizes a thin tube-like structure, with lengths well beyond the critical spacing anticipated without flow. We characterize the shape, length, and stability of this novel “fluid tube structure as a function of the flow rate, ring diameter, and ring shape.

1We acknowledge the financial support of the Cal Poly Physics Department

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A09 FSI: Sheets / Membranes 213 - Shai B. Elbaz, Technion - Israel Institute of Technology
3:00PM A09.00001 Experimental and computational studies of flexible membrane aerodynamics. RODRIGO PADILLA, CONAL THIE, VIBHAV DURGESH, TAO XING, University of Idaho — Fluid-structure interaction (FSI) problems involve the interaction of flexible bodies or membranes like flexible airfoils, parachutes, and sails, with fluid flows. A detailed understanding of the flow dynamics and structure behavior is critical to resolving FSI problems. The objective of the current study is to quantify the impact of critical dimensionless parameters (Re, aspect ratio, dimensionless rigidity) on flow field, vortex shedding, surface pressure distribution, and membrane response, for a canonical rectangular flexible membrane with varying aspect ratios. A collaborative experimental and computational investigation was performed, and the membrane was fabricated using 3D printers. The experiments for this investigation were performed in a subsonic wind tunnel facility and the flow field was measured using Particle Image Velocimetry (PIV). The Reynolds number ranged from 50,000 to 200,000. Analytical and computational fluid dynamics (CFD) studies were performed. The results show that the interaction between the front and rear segments is the dominant mechanism for instability for various discrete locations of clamping. The results yield the maximal stable speed as a function of elastic damping, fluid density and location of clamping. The results show that the interaction between the front and rear segments is the dominant mechanism for instability for various discrete locations of clamping.

3:13PM A09.00002 ABSTRACT WITHDRAWN —

3:26PM A09.00003 Dynamics and instabilities of an arbitrarily clamped elastic sheet in potential flow. SHAI B. ELBAZ, NETHANEL CHEN, AMIR D. GAT, Technion - Israel Institute of Technology — Shape-morphing airfoils have attracted much attention in recent years. They offer substantial drag reduction by comparison to conventional airfoils and a premise of superior aerodynamic performance. In the current work, we model a shape-morphing airfoil as two, rear and front, Euler-Bernoulli beams connected to a rigid support. The setup is contained within a uniform potential flow field and the aerodynamic loads are modeled by thin airfoil theory. The aim is to study the dynamics and stability of such soft shape-morphing configurations and specifically the modes of interaction between the two airfoils against different initial conditions which are validated by numerical calculations based on commercially available software. We then examine stability and transient dynamics by assuming small deflections and applying multiple-scale analysis to obtain a stability condition. The condition is attained via the compatibility equations of the orthogonal spatial modes of the first-order theory. We then examine stability and transient dynamics by assuming small deflections and applying multiple-scale analysis to obtain a stability condition. The condition is attained via the compatibility equations of the orthogonal spatial modes of the first-order theory.

3:39PM A09.00004 Deformations, forces and flow field around a compliant membrane disk. ASIMANSHU DAS, VARGHESE MATHAI, KENNETH BREUER, School of Engineering, Brown University — Highly compliant membranes exhibit large scale deformations and can alter the nature of the flow with which they interact. We focus on understanding the kinematics and dynamics of circular compliant membranes with varying shear modulus, G, placed head-on in a uniform flow. The experiments were carried out in a closed-loop low-speed wind tunnel with Reynolds number in the range of $10^4$ to $10^5$. We measure the average and fluctuating properties of the compliant structure, including its deformation and drag forces. The membrane deforms into parachute-like shapes depending on the value of a dimensionless number - the Aerelastic number, $Ae$ - which measures the relative importance of elastic and aerodynamic stresses. With decreasing $Ae$ (either higher speed, or a softer membrane), the membrane evolves from a flat circular disk-like shape to a nearly hemispherical shape. The drag coefficients vary from 1.1 to 1.4. We provide comparisons with the flow-induced forces on rigid shells of similar shape.

3:52PM A09.00005 Lift and Thrust Measurements on a Flapping Membrane Foil. GALI ALON TZEZANA, VARGHESE MATHAI, KENNETH BREUER, School of Engineering, Brown University — Flapping compliant membrane wings demonstrate increased lift and thrust when compared to rigid wings, particularly when flapping close to the natural frequency of the wing. However, if the wing is too compliant, the forces are decreased. Here, we performed experiments to measure the hydrodynamic forces acting on a heaving compliant membrane in a water flume facility, at a Reynolds number of 20,000-60,000. By varying the membrane’s elastic modulus, thickness and pre-stretch, we modify the natural frequency of the membrane. We observe linear and nonlinear behaviors of the forces and membrane deformations with respect to the Strouhal number and natural frequency. For cases with small deformations, we compare the experimental results with a linear, small amplitude potential flow based model. Finally, we discuss the flow structure and wake patterns obtained from Particle Image Velocimetry (2-D PIV) measurements, and their relation to propulsive performance.

4:05PM A09.00006 Large-amplitude membrane dynamics in inviscid flow. CHRISTIANA MAVROYIAKOMOU, SILAS ALBEN, University of Michigan — We study the dynamics of thin membranes—extensible sheets with negligible bending stiffness—and their behavior in an inviscid background flow. This is a benchmark fluid-structure interaction that has previously been studied mainly in the small-deflection limit, where the flat state may be unstable. Related work includes the shape-morphing of airfoils and bat wings. We study the initial instability and large-amplitude dynamics with respect to three key parameters: membrane mass density, stretching rigidity, and pretension. When both membrane ends are fixed, the membranes become unstable by a divergence instability and converge to steady deflected shapes. With the leading edge fixed and trailing edge free, divergence and/or flutter occurs, and a variety of periodic and aperiodic oscillations are found. With both edges free, the membrane may also translate transverse to the flow, with steady, periodic, or aperiodic trajectories.

4:18PM A09.00007 Efficiency Enhancements in Hydrokinetic Energy Harvesting Achieved Using a Compliant Membrane Hydrofoil. VARGHESE MATHAI, GALI ALON TZEZANA, KENNETH BREUER, School of Engineering, Brown University — Hydrokinetic energy harvesting using an oscillating hydrofoil has recently received increased attention as an alternative to conventional rotary turbines. One of the challenges in the development of commercially viable flapping foil technology is to attain cycle efficiencies comparable to those of rotary turbines. Here we experimentally study the energy harvesting performance of a compliant membrane hydrofoil undergoing heaving and pitching oscillations in a uniform flow. Membrane foils with different properties: elastic modulus, thickness and pre-stretch were fabricated and tested in a water flume facility. The Reynolds number based on the chord length and free stream velocity was $Re = 5 \times 10^4$, and the reduced frequency $f^* \in [0.1, 0.2]$. When compared to a rigid symmetric hydrofoil, the membrane foil is able to dynamically adapt its shape and camber during each oscillation cycle, yielding up to 50% higher lift forces and 30% improvement in cycle efficiency. The flow structure and wake patterns obtained from Particle Image Velocimetry (2-D PIV) measurements will also be discussed.

3:00PM A10.00001 On the Morison equation and heave plate hydrodynamics, CURTIS RUSCH, University of Washington, BENJAMIN MAURER, University of Washington Applied Physics Laboratory, BRIAN POLACYE, University of Washington — Ocean wave energy converters often use a submerged reaction body, such as a heave plate, to generate electricity from wave motion. Analysis of heave plate hydrodynamics typically utilizes the Morison equation to estimate the contributions of drag and inertia to hydrodynamic force, neglecting further components, such as the vortex force discussed extensively by Saripaya. We perform laboratory experiments to evaluate the representativeness of a Morison decomposition over a range of operational and survival states, representing Keulegan-Carpenter (KC) numbers of 0.5 to 4. Driving a hexagonal conic heave plate in regular sinusoidal motion with 17 combinations of period and amplitude, we measure force at the plate and visualize the surrounding flow. Using the Morison equation, we calculate both constant coefficients of drag and added mass and phase-dependent variable coefficients. We find that constant coefficients adequately describe hydrodynamic force at low KC number, but this accuracy decreases with increasing KC number. Variable coefficients accurately reconstruct hydrodynamic forces over the full range of KC numbers investigated. We discuss this discrepancy in the context of the vortex term using dye visualization, and discuss implications for wave energy converter design.

1NSF Graduate Research Fellowship Program and the Naval Facilities Engineering Command

3:13PM A10.00002 Vortex-Induced Vibration and Galloping of a Flexible Square Prism, RONALD COLMON, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — We have studied flow-induced oscillations of a tension-dominated flexible beam with a square cross-section placed perpendicular to the incoming flow. The prism with an aspect ratio of 32 was fixed at its both ends and placed in the test-section of a recirculating water tunnel. The Reynolds number was varied from 400 to 2400. Tracker points were evenly spread along the length of the prism, on two perpendicular sides, in order to measure the prism’s displacements in the crossflow (CF) and inline (IL) directions. Displacements of these points were tracked using two synchronized high-speed cameras. It was found that at low reduced velocities the first and then the second structural modes were excited in the CF direction, together with the second and the fourth modes in the inline direction, resulting in vortex-induced vibration with “figure 8” trajectories. At higher reduced velocities, the amplitude of oscillations increased dramatically, and galloping-type oscillations were observed.

1Spaulding-Smith Diversity STEM Fellowship

3:26PM A10.00003 The formation process of leading and secondary vortices, DIEGO FRANCESCANGELI, KAREN MULLENERS, EPFL — Vortex formation is a limiting process. When a plate is accelerated from rest or fluid is impulsively ejected from a piston-cylinder apparatus, a primary or leading vortex grows in size and strength up to a limit when it pinches-off. Beyond this point, secondary vortices akin to a Kelvin-Helmholtz instability are generated. These secondary vortices are similar to each other but smaller in size and lower in strength than the leading vortex. The motivation to understanding the difference between the leading and secondary vortices, leads us to study the formation process of vortices around a moving rectangular plate in a quiescent fluid. Eulerian and Lagrangian methods are used to track vortices and determine their circulation, vorticity distribution, and size from experimental velocity field data. The experimental data is further compared with different theoretical vortex models to elucidate the differences between the formation process of the leading and the secondary vortices.

3:39PM A10.00004 Dynamics and flow structures in the vortex-induced vibration of a curved flexible cylinder, BANAFSHEH SEYED-AGHAZADEH, Darmouth Campus, University of Massachusetts, BRIDGET BENNER, XHINO GJOKOLLARI, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — Vortex-induced vibration of a curved flexible circular cylinder placed in the test section of a re-circulating water tunnel and fixed at both ends was studied, experimentally. Both the concave and the convex orientations (with respect to the oncoming flow direction) were considered. The cylinder was tension-dominated by its own weight with an aspect ratio of 86 and a low-mass ratio of 3.7. High-speed imaging technique was employed to record the oscillations of the cylinder in the crossflow direction for a reduced velocity range of $Re = 370-2400$. Mono- and multi-frequency responses as well as transition from low mode numbers to high mode numbers are observed. Regardless of the type of curvature, both odd and even mode shapes are excited in the crossflow directions. However, the response of the system, in terms of the excited modes, amplitudes and frequencies of oscillations, is observed to be sensitive to the direction of curvature (concave vs. convex), in particular at higher reduced velocities where mode transition happens. Hydrogen bubble image velocimetry flow visualization was conducted at multiple points along the length of the curved cylinder. Intermittency in the vortex shedding patterns in the wake of the cylinder and alternating wake along the cylinder are observed. The observed altering wake corresponds to the multi-modal excitation and dominant mode transition along the length of the cylinder.

3:52PM A10.00005 Decay of vortex rings in spheroidal confined domains, MILAD SAMAEED, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Recent studies studied the decay of vortex rings in radially confined domains; however, the vortex ring decay process in spheroidal confinement (axial and longitudinal confinement) remains unclear. This type of confinement is observed in the vortex rings formed during filling of the human cardiac left ventricle. We hypothesized that the rate of circulation decay increases in more axially confined domains. A piston-cylinder setup was used to generate vortex rings for this study. 2D time-resolved PIV was used to quantify the flow fields within three different aspect ratios of spheroidal silicone models (0.8, 1, and 1.25) under different filling duration, Reynolds number, and deceleration time. The formation number and peak circulation remained unaffected regardless of model shape. However, for the cases with the same filling duration, the vortex ring pinched-off earlier by shortening the acceleration time. We observed a higher rate of decay for the model with a lower aspect ratio (more axially confined).

1This work was supported by a Carroll M. Leonard Faculty Fellowship to Santhanakrishnan.

4:05PM A10.00006 ABSTRACT WITHDRAWN

4:18PM A10.00007 ABSTRACT WITHDRAWN

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A11 Turbulence: Wall-bounded Flows I 3B - Jason Hearst, NTNU
3:00PM A11.00001 Direct Calculation of Eddy Viscosity of Turbulent Channel Flow

supported by Boeing.

3:13PM A11.00002 A minimal quasilinear approximation of turbulent channel flow

YOUNG-YUN HWANG, Department of Aeronautics, Imperial College London, BRUNO ECKHARDT, Fachbereich Physik, Philips-Universität Marburg — Townsend’s model of attached eddies for boundary layers is analysed within a quasi-linear approximation. The velocity field is decomposed into a mean profile and fluctuations. While the mean is obtained from the nonlinear equations, the fluctuations are modelled by replacing the nonlinear self-interaction terms with an eddy-viscosity-based turbulent diffusion and a stochastic forcing. The colour and amplitude of the stochastic forcing are then determined self-consistently by solving an optimisation problem which minimises the difference between the actual Reynolds shear stresses and the model. When applied to turbulent channel flow in a range of friction Reynolds number from $Re_f = 500$ to $Re_f = 20000$, the resulting turbulence intensity profile and energy spectra exhibit exactly the same qualitative behaviour as DNS data throughout the entire wall-normal location, thereby reproducing the early theoretical predictions of Townsend and Perry within a controlled approximation to the Navier-Stokes equation.

3:26PM A11.00003 Minimal-span direct simulation of transient, accelerating channel flows and application with wall riblets.

SAURABH PARGAL, Student, JUNLIN YUAN, Assistant professor, GILES BRETON, Professor — Minimal-span simulation of wall turbulence is an attractive approach for reducing simulation cost when focusing on near-wall phenomena associated with frictional drag modification due to surface riblets or roughness. We evaluate the capability of such simulations for a periodic smooth-wall channel flow subjected to rapid acceleration from a friction Reynolds number of 180 to one of 420. Compared to a full-span simulation, the single-point statistics, two-point correlations and spectral analyses indicate that the minimal span is sufficient to capture the pseudo-laminarization phenomenon, since the stabilization effect of the increased ensemble-averaged shear is confined to the near-wall region. As the increased shear is relaxed, the retransition to a new equilibrium state, though starting from the wall, is slightly delayed compared to that in a full channel. The wall is then covered with saw-tooth riblets of a uniform height, which is 7.5 wall units at the start of, and 17.5 at the end of acceleration. Lower Reynolds stresses (in wall units) and a slower retransition are observed in the riblet flow, compared to a smooth-wall channel flow. The results demonstrate both advantages and limitations of minimal-span simulations of non-equilibrium wall turbulence.

3:39PM A11.00004 Orr-Sommerfeld and Squire modes in fully-turbulent channel flow

RYAN MCMULLEN, KEVIN ROSENBERG, BEVERLEY MCKEON, Caltech — The Orr-Sommerfeld (OS) and Squire (SQ) operators from classical stability literature have gained renewed interest in the context of linear analyses of turbulent shear flows. We demonstrate that the recently-proposed decomposition of the resolvent operator into two distinct families related to the OS and SQ operators (Rosenberg & McKeon, 2019) results in accurate low-order representations of the second-order statistics in turbulent channel flow. It is shown that the ability to match all components well is due to the isolation of the v response in the OS modes. This enables competition between the OS and SQ vorticity, which is interpreted as a phase difference between the two sets of modes. Additionally, the relative Reynolds number scalings for the two families of resolvent weights are derived for the universal classes of resolvent modes presented by Moarref et al. (2013). Implications for modeling nonlinear interactions in wall-bounded turbulence are discussed.

3:52PM A11.00005 Direct Numerical Simulations of transverse acoustic forcing in fully developed channel flow turbulence

ANDREA GRUBER, SINTEF, MELISSA KOZUL, JAMES R. DAWSON, NTNU — Hydrogen-firing of gas turbines is a promising direction for clean and efficient carbon-free power generation. However, premixed combustion of undiluted hydrogen in low-emission combustion systems can result in unstable operating modes for gas turbines and their development must solve issues posed by flame flashback (off-design upstream propagation), and flame combustion dynamics (coupling of longitudinal or transversal acoustic waves with fluctuating heat release). The causal relationship between longitudinal acoustically-induced bulk flow fluctuations (due to combustion dynamics) and the occurrence flashback in gas turbine burners has been established for some time (Fritz et al., J Eng Gas Turbines Power 2004) while the key role of the near-wall ‘streaks’ of the turbulent boundary layer in controlling the flashback propagation was only explained more recently (Gruber et al., J Fluid Mech 2012). In this context, it is important to examine the effect of transversal acoustic waves, resulting from azimuthal modes resonating in annular combustors, on the characteristic streaky structure of near-wall turbulence. We present DNS results from an idealized channel flow configuration that employs periodic boundary conditions in the streamwise and spanwise directions, where sinusoidal pseudo-acoustic waves are imposed transversally to the bulk flow, mimicking a limit cycle. We report the response of the near-wall streaks of the velocity field to the frequency and amplitude of the imposed acoustic waves.

4:05PM A11.00006 Lifetimes of the large-scale motions in channels up to $Re_f \approx 5000$

MIGUEL ENCINAR, ALBERTO VELA-MARTIN, JAVIER JIMENEZ, Universidad Politecnica de Madrid — The energy spectra of (wall-bounded) turbulent flows is unmistakeably one of the most important tools in turbulent research. Due to past limitations of computational resources, direct numerical simulations have traditionally focused on the computation of the spatial spectra, neglecting its temporal counterpart. We compute the spatio-temporal spectra of the large scales (larger than one fifth of the channel half-height) of channels up to $Re_f \approx 5000$. Using the spectra, it is possible to extend the definition of ‘correlation time’ to provide a meaningful estimation of the lifetime for every spatial wavelength, and at every height. We find the lifetimes to increase with the distance to the wall and with the stream and spanwise wavelengths. There is almost no difference between in the lifetime of a given wavelength between velocity components. Nevertheless, the expected lifetime of each component is different due to the difference in their energy spectra, i.e. the streamwise component lives longer because has more energy in long wavelengths. We also found that the lifetimes scale across Reynolds numbers with the channel half-height and the bulk velocity, instead of the friction velocity.

1Supported by ONR (N00014-17-1-2102) is acknowledged.

1This work was supported by the AFOSR through grant FA 9550-16-1-0361 and ONR through grant N00014-17-1-3022.

1Funded by the ERC COTURB project. Part of the computational time was provided by the PRACE TIER-0 project “TREC”.
4:18PM A11.00007 Turbulent mean flow profile in 2D channel flow. VILDA MARKEVICIUTE, RICH KERSWELL, University of Cambridge — Two-dimensional channel flow is well known to exhibit chaotic behaviour in which structures common to turbulent flows such as vorticity sweeps, large-scale intermittency, and quasi-periodic bursting are observed (Jimenez 1990). Recently, Falkovich and Vladimirova (2018) have found that the time-averaged turbulent mean flow at constant pressure gradient exhibits a remarkably simple wave-like structure which is known to exist for low Reynolds numbers (Re), including below the linear instability threshold (Re=5772). Trying to confirm this finding, here we revisit this flow imposing constant mass flux driving instead. At least in short channels, we find unexpectedly long-lived asymmetric states beyond a critical Reynolds number where the turbulence is confined to one of the channel walls. This poses an interesting question of how long one needs to wait to recover the expected symmetric mean flow or whether there really is bistability of turbulent states centred on one or other of the walls.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A12 Separated Flows: General 303 - Xiaofeng Liu, San Diego State University

3:00PM A12.00001 Bursting of Laminar Separation Bubbles, O. N. RAMESH, ABHIJIT MITRA, Indian Institute of Science, Bangalore — The transition of a small laminar separation bubble (LSB) to a long LSB is called bursting, and this might be abrupt in some airfoils. Bursting is manifested as a significant departure of the surface pressure distribution from its inviscid flow distribution, with a considerable drop in the peak suction pressure. In our present study of flow over Eppler367 at an angle of attack of 8 degrees, we have chosen three representative LSBs: long, transitional and short bubbles. Bursting has been often linked to the onset of absolute instability of the separated shear layer. On impulsively forcing these three bubbles, it is found that all the bubbles are convectively unstable in the high-frequency range. However, all of them, though an order of magnitude lesser in short bubbles, show tendencies of absolute instability in the low-frequency range. A self-limiting behavior for the unforced transitional and long LSBs is evident. An explosive growth of high-frequency activity is found to be strongly correlated to the low-frequency flapping motion of the longer bubbles. Negative production of turbulent kinetic energy is responsible for the runaway effect observed during bursting.

3:13PM A12.00002 Sudden kinks in proper orthogonal decomposition modes of vorticity field for a turbulent flow past a ship topside subject to oscillatory motion. XIAOFENG LIU, San Diego State University, ANISH SYNDAY, NAPEI BI, Naval Surface Warfare Center Carderock Division, SAN DIEGO STATE UNIVERSITY COLLABORATION, NAVAL SURFACE WARFARE CENTER CARDEROCK DIVISION COLLABORATION — Complex flow fields above and around ship deck often exhibit a wide range of temporal and spatial features in fluid motion, which affects not only the ship performance, but also Launch and Recovery (L&R) of air vehicles. To characterize the complex turbulent flow over a ship topside, Time-Resolved Particle Image Velocimetry (TR-PIV) is used in a subsonic wind tunnel test with both stationary and oscillatory modes of the ship model. The tests are conducted at a wind speed of 10.3 m/s and ship oscillations at a frequency of 2.0 Hz with an amplitude of 2.5 degrees. Proper Orthogonal Decomposition (POD) analysis, capable of extracting energetically and dynamically significant features of complex fluid flows, is applied to the time series of the planar velocity and vorticity distributions obtained by the PIV measurements. The POD analyses show that a novel phenomenon featuring significant sudden discontinuities in the vorticity POD mode pattern distributions dubbed as “kinks” is observed for the test model subject to oscillatory motion. In contrast, such kinks are not found in the vorticity POD modes for the test case without the model oscillatory motion, nor the velocity POD modes for both the stationary and the oscillatory tests.

3:26PM A12.00003 Dynamics of the shear layers detaching from the upstream corners of an elongated rectangular cylinder, MARIA VITTORIA SALVETTI, ALESSANDRO MARIOTTI, ELENA PASQUALETTO, BENEDETTO ROCCHIO, DICI, University of Pisa — The behavior and the dynamics of the shear layers detaching from a rectangular cylinder, having chord-to-depth ratio equal to 5, are investigated through highly-resolved large-eddy simulations and experimental velocity and pressure measurements. The considered configuration, which is the object of the benchmark BARC, is characterized by flow reattachment on the cylinder sides in the rear part of the body. It is shown that the mean length of the separated region, and hence the velocity and pressure measurements. The considered configuration, which is the object of the benchmark BARC, is characterized by flow reattachment on the cylinder sides in the rear part of the body. It is shown that the mean length of the separated region, and hence the shear-layer transition length. The characteristic frequencies of the shear-layer dynamics are also investigated and compared with those characterizing the vortex shedding in the wake and the unsteady aerodynamics loads. Finally, the effects of small rounding of the upstream corners on the previously mentioned quantities and phenomena are briefly analyzed.

3:39PM A12.00004 How dynamic is static stall? Julien DEPARDAY, KAREN MULLENNERS, UNFoLD, Institute of Mechanical Engineering, Ecole Polytechnique Federale de Lausanne, CH-1015 Lausanne, Switzerland — The static stall angle is the critical angle of attack above which the flow detaches from the airfoil’s surface. When the static stall angle is exceeded, the transition from attached to separated flow is not instantaneous. The transient development is characterized by different stages and time-scales similar to the dynamic stall flow development stages and stall delays. Static stall can be seen as an extreme case of dynamic stall where there is no motion during the stall development. On a NACA0018 profile, we initiated static stall by an increase in the angle of attack from 0.3° to 0.3° above the static stall angle in 10% of a convective time unit. We measured the time-resolved flow field and aerodynamic forces and analysed their temporal evolution. The transition time from attached to separated flow is longer in the static stall case than in the dynamic stall cases even though the topological flow development is comparable. Special emphasis is directed towards linking the inception of stall to the motion of the leading edge stagnation point. These results will serve as a basis to improve low-order models based on indicial response.

1Supported by the ONR Summer Faculty Research Program

1This work was supported by the Swiss national science foundation under grant number PYAPP2 173652
3:52PM A12.00005 Role of an active trailing-edge flap of a pitching airfoil undergoing dynamic stall1, GUOSHENG HE, EPFL, LARS SIEGEL, ARNE HENNING, DLR, KAREN MULLENNER, EPFL — The flow around a pitching NACA0015 airfoil with an active trailing-edge flap is investigated using two-dimensional time-resolved particle image velocimetry and surface pressure measurements. The Reynolds number based on the chord length is about $5.5 \times 10^5$. The airfoil is pitching around the static stall angle of attack of $20^\circ$ and the flap is either oscillating around the symmetrical plane at $\beta = 0^\circ$ or fixed at a constant deflection angle. The pitching of the airfoil and the deflection of the flap can be individually controlled in terms of mean angle, oscillating amplitude, frequency and initial phase angle. Below the stall angle, the lift of the static airfoil increases proportionally with the increase of flap deflection angle in the investigated range of $-20^\circ \leq \beta < 20^\circ$. Variations of the phase delay between the oscillations of the main airfoil and the flap lead to a rotation, expansion, or contraction of the dynamic stall lift curves. Higher-order harmonic flap oscillations for the pitching airfoil result in bending or twisting of the lift curves. Quantitative evidence has been extracted from the PIV data to help elucidate the modified aerodynamic characteristics of the pitching airfoil manipulated by the active flap.

1DFG-SNSF grant number 20021E-169841

4:05PM A12.00006 Enhancement of Slat Airfoil Configuration using Invasive Weed Optimization Framework coupled with Artificial Neural Networks,1, SUSHRUT KUMAR, PRIYAM GUPTA, RAJ KUMAR SINGH, Delhi Technological University — This research attempts to develop an optimization scheme by integrating Genetic Algorithm and Artificial Neural Network (ANN) - surrogate model which was successfully implemented to optimize Leading Edge Slat shape and configuration. The optimization model used Invasive Weed Optimization. Bezier curves were used as an aerodynamic shape parameterization method to ensure the generation of smooth-contoured slat profiles. Geometry, Overhang, Depth and Deflection were taken as the defining parameters for each individual. These parameters were varied within a range which is 3.4% to 15% of slat chord for Overhang, -5% to 4% of slat chord for Depth, -5% to 5% of initial deflection for Deflection and the ordinate of shape control points had a variation of $\gamma$ to $\gamma$3. The standard deviation decreases non linearly from 6 to 0.005 with a modulation index of 3. Multiple Computational Fluid Dynamics simulations were run for each individual under various operating conditions to evaluate their fitness (lift to drag ratio). The data generated from this process was used as training and test sets for the ANN. Shape control points, angle of attack, Reynolds number and operating conditions were taken as input parameters for the neural network to predict lift to drag ratio. The developed technique showed approximately 85% improvement in the time taken and allowed algorithm to better explore design space.

1We acknowledge Fluid Systems Lab, Delhi Technological University for providing access to the computing facility for performing the simulations.

4:18PM A12.00007 Resolvent analysis of laminar separation bubble over an airfoil at high Reynolds number using discounting and randomized linear algebra1, CHI-AN YEH, University of California, Los Angeles, STUART BENTON, Air Force Research Laboratory, Wright-Patterson Air Force Base, KUNIHIKO TAIRA, University of California, Los Angeles, DANIEL GARMANN, Air Force Research Laboratory, Wright-Patterson Air Force Base — We perform resolvent analysis to examine the perturbation dynamics over the laminar separation bubble (LSB) that forms near the leading edge of an airfoil at Re=5E5. While we focus on the LSB residing over 65% of the chord, the resolvent operator is constructed about the global mean flow over the airfoil, avoiding issues due to domain truncation. Moreover, randomized SVD is adopted in the analysis to relieve the computational cost for the high-Re global baseflow. To examine the local physics over the LSB, we consider the use of exponential discounting to limit the time horizon that allows for the instability to advect along the baseflow. In addition to discounting, we also examine the use of a spatial mask that highlights the LSB in the analysis. We find that discounting is required to reveal the stability characteristics of the LSB due to the unstable baseflow, regardless of the use of the mask. With discounting, the gain distribution over frequency accurately captures the spectral content over the LSB obtained from LES. The peak frequency also agrees with previous flow control results on suppressing dynamic stall. We show that discounted analysis is the appropriate approach for unstable baseflow and that randomized approach can expand global resolvent analysis to high-Re flows.

1This study is supported by Air Force Office of Scientific Research and Office of Naval Research.

Saturday, November 23, 2019 3:00PM - 4:05PM – Session A13 Magnetohydrodynamics

3:00PM A13.00001 Inclined turbulent thermal convection in liquid sodium1, LUKAS ZWIRNER, Max Planck Institute for Dynamics and Self-Organization, RUSLAN KHALILOV, ILYA KOLESNICHENKO, ANDREY MAMYKIN, SERGEY MANDRYKIN, ALEXANDER PAVLINOV, ALEXANDER SHESTAKOV, ANDREY TEIMURAZOV, PETER FRICK, Institute of Continuous Media Mechanics, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization — Inclined turbulent thermal convection at large Rayleigh numbers (Ra) in extremely small Prandtl-number (Pr) fluids is studied by measurements and high-resolution numerical simulations. The working fluid is liquid sodium (Pr about 0.0094) and the considered Ra is around $1.5 \times 10^7$. The convection cell is a cylinder with equal height and diameter, where one circular surface is heated and another one cooled. For the limiting inclination angle $\beta$, which correspond to Rayleigh-Benard convection ($\beta = 0$) and vertical convection ($\beta = \pi/2$), the scaling relations of the mean heat flux (Nusselt number Nu) with Ra are studied. Any inclination of the RBC cell leads to an increase of Nu; the maximal Nu is obtained, however, for a certain intermediate value of $\beta$. For small $\beta$, the large-scale circulation (LSC) exhibits a complex dynamics, with torsion and sloshing modes, which are suppressed for large $\beta$. When the LSC is twisted, the volume-average vertical heat flux is minimal, and it is maximal, when the LSC sloshing brings together the hot and cold streams of the LSC.

1The work is supported by the Deutsche Forschungsgemeinschaft (DFG), grants Sh405/7 (SPP 1881 Turbulent Superstructures) and Sh405/4 (Heisenberg fellowship) and the Leibniz Supercomputing Centre.
3:13PM A13.00002 Redescribing of Busse balloon on Rayleigh-Bénard convection imposed by horizontal magnetic field  
YUJI TASAKA, Hokkaido University, TAKATOSHI YANAGISAWA, JAMSTEC, TOBIAS VOGT, SVEN ECKERT, HZDR — Laboratory experiment of RBC was performed using GaInSn eutectic as a typical low Pr fluid with quasi-uniform horizontal magnetic field to elucidate how the Busse balloon describing stability of 2D rolls is modified by Lorenz force effect. Development of the flow from steady rolls to unsteady state were investigated with decreasing Chandrasekhar number $Q$ in a range, $2.5 \times 10^{-8} \leq Q \leq 1.9 \times 10^{-7}$ at fixed $Ra$ numbers, $7.9 \times 10^4 \leq Ra \leq 1.8 \times 10^5$, by ultrasonic velocity profiling. The velocity profile measurements showed the dynamic morphology of the oscillatory convection, 2D oscillation observed at the onset of oscillations, oscillations of recirculation vortex pairs between the main rolls, and synchronous motion of the main rolls. The measurements also suggested that the oscillation occurs at similar Reynolds numbers $Re \approx 900$ and may be caused by instabilities on the recirculation vortex pair. This finding suggests that the oscillations are essentially different from generally observed traveling waves described as the oscillatory instability considered in the Busse balloon. Power law found on the variation of $Re$ with $Q$ suggested the 2D oscillation is dominated by ratio between side wall Hartmann braking and buoyancy.

3:26PM A13.00003 Heat transfer and flow regimes in quasi-static magnetoconvection with a vertical magnetic field  
MING YAN, MICHAEL CALKINS, STEFANO MAFFEI, KEITH JULIEN, University of Colorado, Boulder , STEVEN TOBIAS, University of Leeds , PHILIPPE MARTI, ETH Zurich — Numerical simulations of Rayleigh-Bénard convection with an imposed vertical magnetic field are carried out over a broad range of Rayleigh numbers and magnetic field strengths. Three magnetoconvection regimes are identified: two of the regimes are magnetically-constrained in the sense that a leading-order balance exists between the Lorentz and buoyancy forces, whereas the third regime is characterized by unbalanced dynamics that is similar to non-magnetic convection. Each regime is distinguished by flow morphology, momentum and heat equation balances, and heat transport behavior. One of the magnetically-constrained regimes appears to represent an ‘ultimate’ magnetoconvection regime in the dual limit of asymptotically-large buoyancy forcing and magnetic field strength; this regime is characterized by an interconnected network of anisotropic, spatially-localized fluid columns aligned with the direction of the imposed magnetic field that remain quasi-laminar despite having large flow speeds. Heat transport is controlled primarily by the thermal boundary layer. Empirically, the scaling of the heat transport and flow speeds appear to be independent of the thermal Prandtl number within the magnetically-constrained regimes.

3:39PM A13.00004 Response of the Free Surface of an Electrically Conducting Liquid to a Magnetic Field  
SURESH MURUGAIYAN, COLIN ADAMS, BHUVANA SRINIVASAN, STEFANO BRIZZOLARA, Kevin T. Crofton Department of Aerospace and Ocean Engineering, Virginia Polytechnic Institute and State University — Response of the free surface of an electrically conducting liquid to a magnetic field that varies in space and time, is studied using numerical simulations. A fully implicit finite volume solver is used to solve magnetohydrodynamic equations. The equations are solved using a segregated approach on a collocated grid arrangement where all the variables are stored at the cell centers. The PISO (Pressure-Implicit with Splitting of Operators) algorithm is used to solve the equation of fluid momentum with the Rhie and Chow momentum interpolation technique to overcome oscillations in pressure. The equation of magnetic field evolution is solved in the same manner as the equation of fluid momentum by introducing an artificial pressure term. The method of Volume Of Fluid (VOF) is used to track the interface between the two immiscible fluids. Numerical experiments concerning the effect of high density ratio on a two-fluid magnetohydrodynamic system shall be carried out.

3:52PM A13.00005 Instability in Electromagnetically Driven Flow between Concentric Spheres1  
SAUL PIEDRA, CONACYT-CIDESI Centro Nacional de Tecnologías Aeronáuticas, ALDO FIGUEROA, IVAN RIVERA, CONACYT-Centro de Investigacion en Ciencias, Universidad Autonoma de Estado de Morelos, PHYSICS OF CONTINUOUS MEDIA LABORATORY TEAM — The rotational flow continuously driven by electromagnetic forcing of an electrolytic fluid in the gap of concentric spheres set-up is studied experimentally and theoretically. The driving Lorentz force is generated by the interaction of a DC electric current radially injected and the dipolar magnetic field produced by a permanent magnet. Velocity profiles in the equatorial plane were obtained using Particle Image Velocimetry (PIV), whereas the radial velocity component of the flow was recorded with Ultrasonic Doppler Velocimetry (UDV). A fully three-dimensional numerical model that takes into account the dipolar magnetic field and the radial dependency of the applied current was developed. The model reproduces the main features of the electromagnetically forced flow. For small injected currents, a quite axisymmetric equatorial recirculation formed mainly by diffusive momentum transport was found. For currents above 200 mA, which corresponds to a Re > 1340, instabilities of the inner boundary layer are observed and the flow becomes three-dimensional and time dependent.

1This work is supported by CONACyT, México, under project 258623. A. Figueroa and S. Piedra thank to Cátedras program from CONACyT.

Saturday, November 23, 2019 3:00PM - 4:31PM —
Session A14 Convection and Buoyancy-driven Flows: Multiphase 307/308 - Raymond Shaw, Michigan Technological University

3:00PM A14.00001 Heat flux in moist and cloudy Rayleigh-Bénard convection1  
RAYMOND SHAW, SUBIN THOMAS, PRASANTH PRABHAKARAN, WILL CANTRELL, Michigan Technological University — Heat flux in moist and cloudy Rayleigh-Bénard convection consists of both sensible heat flux (temperature flux) and latent heat flux (water vapor flux and condensation/evaporation growth of cloud droplets). Under cloudy conditions, condensation of water vapor away from the boundaries is a source term for temperature and a sink term for water vapor. Thus, the condensation of water vapor reduces the mean water vapor concentration in the bulk and increases the mean bulk temperature. This in turn enhances the latent heat flux and lowers the sensible heat flux passing through the system. We investigate the effect of condensation rates in a cloudy Rayleigh-Bénard convection system, using an experimentally validated LES model. We propose a modified heat flux that remains conserved under these conditions, and we explore the influence of varying cloud properties such as cloud droplet concentration (modulated via the aerosol properties) and liquid water mixing ratio.

1This work is supported by NSF grant AGS-1754244.
3:13PM A14.00002 Effects of the Large-Scale Circulation on Temperature and Water Vapor Distributions in the Michigan Tech II-Chamber

JESSE ANDERSON, GREGORY KINNEY, PRASANTH PRABHAKARAN, SUBIN THOMAS, RAYMOND SHAW, WILL CANTRELL, Michigan Technological University — In Rayleigh-Bénard convection, it is well known that within the turbulent motion a mean flow forms, commonly referred to as the large-scale circulation. We report experimental results on the nature of this circulation and its impact on the temperature and water vapor fields in Michigan Tech’s II-chamber (Aspect ratio=2 and Pr = 0.7) under dry, moist (no injection of aerosols) and cloudy conditions. The II-chamber is designed to study aerosol-cloud interactions in a turbulent environment. These interactions are strongly influenced by the temperature and water vapor fields because they control the growth rates of each cloud droplet. The differential growth rates between droplets could result in a broadening of the cloud droplet distribution which is important for the onset of precipitation in clouds. We report various features of the circulation - an azimuthal oscillation, a sloshing mode, and a possible compression along the axis of the main roll. In addition, the temperature and water vapor concentration were measured and analyzed with respect to the orientation to the large-scale circulation. The distributions were found to be positively skewed along the updraft and negatively skewed along the downdraft.

1National Science Foundation, AGS 1754244

3:26PM A14.00003 A Lagrangian network-based analysis of evaporative droplets in Rayleigh-Bénard convection

THEO MACMILLAN, DAVID RICHTER, University of Notre Dame — Rayleigh-Bénard convection is characterized by different scales of coherent motion, from a large-scale circulation (LSC) to transient thermal plumes. To better understand the mixing behavior of evaporative droplets in such a system, we assemble a time-evolving weighted network from the droplets’ individual trajectories using direct numerical simulation (DNS) with Lagrangian droplet tracking. By varying the time over which the network is assembled, we detect both the LSC and transient thermal plumes in the spectral gap of the network’s Laplacian while accurately determining their respective time scales. Using tools developed for the analysis of complex graphs, we are able to objectively characterize the limiting cases (i.e., fast microphysics dominated and slow microphysics dominated mixing) and the interplay between short time-scale coherent structures and the LSC in the dynamics of the droplet ensemble. This has implications for understanding, for example, essential mechanisms of the formation of rain, including cloud entrainment and droplet spectra broadening processes such as stochastic condensation.

1ONR Grant N00014-16-1-2472 and NSF Grant AGS-1358593

3:39PM A14.00004 Dynamics of air bubbles in Rayleigh-Bénard Convection: pair dispersion and effect of initial separation

LEONARDO P CHAMORRO, JIN-TAE KIM, University of Illinois at Urbana-Champaign, JAEWOOK NAM, Yonsei University, SHIKUN SHEN, University of Illinois at Urbana-Champaign, CHANGHOON LEE, Yonsei University — Laboratory experiments were performed to uncover the dynamics of bubbles in Rayleigh-Bénard (RB) convection at Ra=1.1x10^10, where streams of 1-mm bubbles were released at various locations from the bottom of the RB tank along the path of the roll structure. 3D particle tracking velocimetry was used to track simultaneously a relatively large number of bubbles, and to quantify the pair dispersion for various initial separations in the range of 25 ≤ η ≤ 225, where η is the local Kolmogorov length scale. Numerical simulation was carried out to further study the role of the bubble’s path instability. Results show that the pair dispersion underwent a transition phase similar to the ballistic-to-diffusive (t^2-to-t) regime in the vicinity of the cell center; however, it approached to a bulk behavior t^1.5 in the diffusive regime as the distance away from the cell center increased. At small initial separation, the pair dispersion exhibited t^1 in the diffusive regime, indicating that the convective turbulence reduced the amplitude of the bubble’s path instability. At large initial separations, the pair dispersion exhibited t^2, showing the effect of the roll structure.

3:52PM A14.00005 Formation of vortex rings and drops at particle-laden fronts in thermally stratified environments

CHEN-YEN HUNG, YI-JU CHOU, National Taiwan University — We conduct numerical simulations to investigate the formation of drops and vortex rings at particle-laden fronts descending in density stratified environments. We show that the temporal evolution can be divided into double diffusion, acceleration, and deceleration phases. The acceleration phase is a result of the vanishing temperature perturbation in the drop during the descent in the layer of uniform temperature. The drop decelerates because it transforms into a vortex ring, whose motion follows the similarity assumption. A theoretical drag model is presented to predict the spherical drop speed with the low drop Reynolds number. In conjunction with the similarity argument for the motion of the vortex ring, our drag model shows good agreement in predicting the drag coefficient for the drop after the drop becomes spherical. Comparison of our drag model with simulations under various bulk conditions and previous experimental results shows good model predictability for the descending speed of drops.

4:05PM A14.00006 ABSTRACT WITHDRAWN

4:18PM A14.00007 Self-sustained biphasic catalytic particle turbulence

ZIQI WANG, Tsinghua University, VARGHESE MATHAI, Brown University, CHAO SUN, Tsinghua University — Turbulence is known for its ability to vigorously mix fluid and transport heat. While over a century of research for enhancing heat transport, few have exceeded the inherent limits posed by turbulent-mixing. Here we have conceptualized a kind of “active particle” turbulence machine: we find that by adding a minute payload of heavy liquid (1% hydrofluoroether) to a water-based turbulent convection system, remarkably, high efficient biphasic dynamics is born, which supersedes turbulent heat transport by up to 500%. The system is unique in that it operates on a self-sustained regime as the distance away from the cell center increased. This work was supported by Natural Science Foundation of China under grant nos. 91852202, 11861131005, 11672156.: 1Department of Energy and Power Engineering, Tsinghua University 2School of Engineering, Brown University 3Department of Energy and Power Engineering, Tsinghua University
In the near field has a suppressive effect on shear layer growth as well as the development of the counter-rotating vortex pair (CVP) and this behavior of Reacting Jet in Crossflow systems can be studied by considering the behavior of coherent structures and their formation and growth.

These experimentally determined coefficients can be used to refine Reynolds-Stress models of atmospheric flows in the presence of a volcanic plume. Currently with Exponent

GREG SAKRADSE, GREGORY SAKRADSE, Portland State University, STEPHEN SOLOVITZ, Washington State University Vancouver, RAUL BAYOAN CAL, Portland State University — Prediction of the development of jets in cross-flow has many useful applications in various fields such as chemical mixing and volcanic plume models. Most models used to predict the growth assume irrotational flow when both atmospheric and high Reynolds number flows are not. Simple laboratory experiments of air jets in crossflow downstream of an active grid system are used to simulate uniform turbulent inflows. A jet of air is injected orthogonally into a closed-loop wind tunnel with several cross-flow velocities and high Reynolds number. Results show that the jet has multiple pathways to instability; non-modal interactions are examined on their multiple time-scales. Helical actuations of the transverse modes, as well as axisymmetric and non-axisymmetric modes is studied. Adjoint modes show that the upstream shear-layer is most sensitive along the upstream side of the jet. High sensitivity downstream modes are sensitive in the cross-flow boundary layer. The transverse jet has multiple pathways to instability; non-modal interactions are examined on their multiple time-scales. Helical actuations of the transverse jet is being studied; these results will be discussed.

Supported by AFOSR.

DNS, stability and sensitivity of the low-speed transverse jet, SAMANTHA HAREL, KRISHNAN MAHESH, University of Minnesota — DNS, tri-global stability and adjoint analysis are performed for a jet in cross-flow (JICF) using a novel capability developed on unstructured grids and parallel platforms. Linear stability analysis reveals that upstream shear-layer modes have frequencies that match simulation and experiment. The relative importance of upstream and downstream shear layer modes, as well as axisymmetric and non-axisymmetric modes is studied. Snapshots are used to visualize the jetties of the jet and describe the trajectory and shear layer expansion are quantified and compared. Analysis is then focused on the evaluation of the dissipation and eddy viscosity for the purpose of refining $k-\epsilon$ modeling parameters. Identifying the differences turbulent cross-flow has on development of the jet and lee-side wake region provides more details on the important features necessary for accurate prediction of the spread and entrainment.

Supported by AFOSR.

3:26PM A15.0003 The role of turbulent inflow on the development of a jet in cross-flow

GRAHAM FREEDLAND, GREG SAKRADSE, Portland State University, STEPHEN SOLOVITZ, Washington State University, RAUL BAYOAN CAL, Portland State University — Prediction of the development of jets in cross-flow has many useful applications in various fields such as chemical mixing and volcanic plume models. Most models used to predict the growth assume irrotational flow when both atmospheric and high Reynolds number flows are not. Simple laboratory experiments of air jets in crossflow downstream of an active grid system are used to simulate uniform turbulent inflows. A jet of air is injected orthogonally into a closed-loop wind tunnel with several cross-flow velocities and three active grid settings to simulate a range of turbulent conditions. Mean flow statistics and Reynolds stresses are computed using PIV data to identify key regions of interaction. The flow-fields are reoriented to provide components normal and tangent to the centerline of the jet and descriptions of the trajectory and shear layer expansion are quantified and compared. Analysis is then focused on the evaluation of the dissipation and eddy viscosity for the purpose of refining $k-\epsilon$ modeling parameters. Identifying the differences turbulent cross-flow has on development of the jet and lee-side wake region provides more details on the important features necessary for accurate prediction of the spread and entrainment.

This work was supported by NSF-EAR-1346580

3:39PM A15.00004 The spatial evolution of anisotropy along a jet in cross-flow

GREGORY SAKRADSE, GRAHAM FREEDLAND, RAUL BAYOAN CAL, Portland State University, STEPHEN SOLOVITZ, Washington State University Vancouver, Volcanic plumes present a unique challenge in the realm of atmospheric flow modelling, and many current models rely on general parameters applied to a wide range of flows. Improvements in modelling can be made by tuning model parameters to specific flows with experimental data. In laboratory conditions a volcanic plume can be approximated as a buoyant jet interacting with a cross-flow. The present work examines a round jet emitted into a cross-flow. Stereo PIV is used to directly measure three components of velocity in the near to medium field of the jet on a plane parallel to the cross-flow and on the central axis of the jet. Buoyant, neutrally buoyant and negatively buoyant cases are considered. The state of anisotropy is determined along lines parallel to the outer edges of the jet shear layers, and the progression of of the state of anisotropy is examined on an Anisotropy Invariant Map (AIM). The development in space of the state of anisotropy is used to refine coefficients in both linear and non-linear return to anisotropy models for the transport of the anisotropy tensor. These experimentally determined coefficients can be used to refine Reynolds-Stress models of atmospheric flows in the presence of a volcanic plume.

3:52PM A15.00005 Flame Position-Shear Layer Offset Effects on Jet in Crossflow Dynamics

VEDANTH NAIR, VISHAL ACHARYA, TIM LIEUWEN, Georgia Institute of Technology — The dynamics and mixing behavior of Reacting Jet in Crossflow systems can be studied by considering the behavior of coherent structures and their formation and growth. This study aims to understand the influence of heat-release on the growth of shear layer vortices (SLV) for a non-premixed, reacting jet in crossflow (RJICF) configuration. Two reacting cases are considered: pure methane jet into a crossflow of air (flame lies outside the jet shear layer) and a diluted methane jet into a crossflow of oxygen (flame lies inside), in order to gauge the influence of changing the location of heat release with respect to the shear layer. These two configurations lead to significant differences in the spatial orientation of shear-heat release interactions and how the vorticity interacts with regions on the other side of the jet. The results demonstrate that the presence of heat release in the near field has a suppressive effect on shear layer growth as well as the development of the counter-rotating vortex pair (CVP) and this effect is amplified in the case where the flame is moved inside the shear layer.

1National Science Foundation (contract # 1705649), contract monitor Harsha Chelliah.
4:05PM A15.00006 Lagrangian Coherent Structures (LCS) in crossflow jets subject to very strong Favorable Pressure Gradient (FPG)¹, GERMAN SALTAR RIVERA, CHRISTIAN LAGARES, GUILLERMO ARAYA, University of Puerto Rico - Mayaguez — Incompressible jets transversely issuing into a spatially-developing turbulent boundary layer is one of the most challenging types of three dimensional flows due to its thermal-fluid complexity and technological applications: film cooling of turbine blades, fuel injection, thrust vector control, chimney plumes, among others. Complex interactions between the jet and the crossflow create a variety of coherent structures which govern the flow’s transport properties, most notably, the counter-rotating vortex pair (CVP). The CVP’s influence in the thermal transport of a turbulent round jet in a crossflow with a strong FPG has been previously studied by Quinones & Araya (2017). The FPG exhibited a damping effect on the CVP wake development and a wall-normal stretch on its geometry. To expand upon this work, we conduct Direct Numerical Simulation (DNS) at different jet velocity ratios (i.e., VR = 0.5, 1 and 2) and make use of the Finite-Time Lyapunov Exponent (FTLE) as well as the Finite-Space Lyapunov Exponent (FSLE) to detect and evaluate LCS. The main purpose is to shed light of the combined effect of crossflow-jet and strong FPG on passive scalar transport.

¹Supported by the Office of Naval Research through the USNA Trident Scholar Program

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A16 Advances in LES Modeling I 4c3 - Sanjeeb Bose, Stanford

3:00PM A16.00001 Towards large-eddy simulation of maneuvering vehicles using an unstructured overset grid method¹, NICK MORSE, WYATT HORNE, KRISHNAN MAHESH, University of Minnesota — We discuss the development of large-eddy simulation (LES) capability towards the simulation of maneuvering vehicles using the novel unstructured overset numerical algorithm developed by Horne and Mahesh [J. Comput. Phys. (2019) 376:585-596]. This method addresses the discrete conservation and scaling challenges of overset methodologies and allows for simulation of flows around complex moving bodies undergoing six degree-of-freedom (6 DOF) motion. We discuss the application of this method to two problems of interest. First, simulations are performed of flow over an axisymmetric body of revolution with comparison to experimental data and previous LES studies. Second, a controller is integrated with the overset methods 6 DOF solver to control the motion of the body in external fluid flow. Validation examples and applications are discussed.

¹This work is supported by the Office of Naval Research.

3:13PM A16.00002 Wall-modeled LES of rough-wall flows using the amplitude modulation framework¹, SICONG WU, University of Illinois at Urbana-Champaign, KENNETH CHRISTENSEN, University of Notre Dame, PAUL FISCHER, University of Illinois at Urbana-Champaign, CARLOS PANTANO, University of Southern California — The framework of amplitude modulation (AM) is ideally suited for the wall-modeled LES of turbulent boundary layer flows since the only model input is the large-scale flow information in the outer resolved region by LES. Adapted from the predictive models pioneered by Mathis et al. (2011) and Mathis et al. (2013), wall-boundary conditions (velocities and wall shear stresses) at the first wall-normal grid point maintained in the log region are proposed for turbulent flow over rough walls, where the model coefficients are calibrated from available DNS and experiments and the universal signals are generated using synthetic turbulence of specified energy spectrum and Reynolds stresses. Statistics including the mean velocity, turbulent stresses and dissipation will be collected and compared to the reference data.

3:26PM A16.00003 Dynamic slip wall model for compressible turbulent flows¹, KEVIN GRIFFIN, Center for Turbulence Research, Stanford University, SANJEEBBOSE, Cascade Technologies, Inc., PARVIZ MOIN, Center for Turbulence Research, Stanford University — The dynamic slip wall model (Bose and Park, ARFM, 2018) for LES is extended to compressible flows. Slip boundary conditions are used for velocity and temperature, and the slip lengths are computed dynamically. The slip wall boundary condition is a new paradigm for wall modeled LES that does not rely on RANS modeling used in traditional wall modeled LES. In this presentation, the slip wall model is applied to a transonic, transitional turbine inlet guide vane (Arts and de Rouvroit, ASME J. Turbomachinery, 1992), using unstructured CharLES code. The slip wall model is applied uniformly to the laminar, transitional, and turbulent sections of the flow. No a priori knowledge of the transition location is needed. The heat transfer on the blade is well predicted using the slip wall model. The use of a slip wall model in the under-resolved laminar section is motivated with an analysis of how slip improves the numerical prediction of the self-similar Falkner-Skan equation with limited resolution.

¹NDSEG, Stanford Graduate Fellowship, NASA
3:39PM A16.00004 Numerical Dissipation and Subgrid-scale Modeling for High-order DG Schemes, DAVID FLAD, USRA, SCOTT MURMAN, NASA Ames Research Center — Using high-order discontinuous-Galerkin (DG) methods for large-eddy simulation (LES) continues to increase in popularity. The methods are often favored over high-order finite-difference schemes due to their geometrical flexibility, dense numerical operators, and minimal inter-processor communication. Often the methods are used with no explicit LES model added, relying on the inherent dissipation of the method to account for unresolved stresses, i.e. implicit LES. In order to obtain nonlinear stability polynomial de-aliasing (“exact” numerical quadrature) is frequently applied. In the current work, we analyze the quality of LES for an high-order implicit time discretization and various high-order spatial discretizations, with a focus on the different effects of inherent numerical dissipation of the schemes. Dissipation-free spatial operators are used to separate modeling terms and discretization errors. The suitability and limitation for high Reynolds number turbulent flows is analyzed and compared to results when adding an explicit subgrid-scale model, such as the Smagorinsky model. Results are presented for decaying homogeneous isotropic turbulence and wall-bounded channel flow.

3:52PM A16.00005 SGS Model Based on Removal of Small-Scale Energy Production through Nonlinear Interactions1, JULIAN DOMARADZKI, GUANGRUI SUN, University of Southern California — Nonlinear dispersive SGS models are not sufficiently dissipative in actual LES. In this work a new nonlinear model is proposed to remove the unphysical energy accumulation near the LES cutoff. The model is derived based on the analysis of the nonlinear energy transfer among scales of different size and can be regarded as a direct removal of the energy production in targeted regions. We compare the present model with other nonlinear models and regularization techniques both theoretically and numerically. We show that through the removal of energy production in a targeted region of scales in the vicinity of the LES cutoff the new model is able to provide sufficient SGS dissipation in actual LES. The scale separation is facilitated by a smooth low-pass filter, which becomes increasingly more active for smaller resolved scales. Since the filter already takes grid size into account, the model is found to consistently produce accurate results in a posteriori tests in LES of turbulent channel flow at various grid resolutions and Reynolds numbers. Our results demonstrate that the energy pileup at small resolved scales in insufficiently dissipative LES can be removed by a simple modification of the nonlinear term without a need for extra dissipative terms such as an eddy-velocity model.

1Supported by NSF

4:05PM A16.00006 An algebraic non-equilibrium wall-stress model for LES by analytically integrating the TBLE1, KAZUHIKO SUGA, TOMOKI SAKAMOTO, YUSUKE KUWATA, Osaka Prefecture University — An algebraic non-equilibrium wall-stress model for large eddy simulation is proposed. The ordinary differential equation (ODE) of the thin-layer approximated momentum equation including the temporal, convection, and pressure gradient terms is considered to form the wall-stress model. Applying the ideas of the analytical wall function for Reynolds averaged turbulence models, the profile of the sub-grid-scale eddy viscosity inside the wall adjacent cells is modeled as two-segment piecewise linear variations. This simplification makes it possible to analytically integrate the ODE near the wall to algebraically give the wall shear stress as the wall boundary condition for the momentum equation. By applying such an integration to the wall-normal velocity component, the methodology to avoid the log-layer mismatch is also proposed. Coupled with the standard Smagorinsky model, the proposed model shows good performance in turbulent channel flows at Re = 1000 - 5000 irrespective of the grid resolutions. The proposed model is also confirmed to be superior to the traditional equilibrium wall stress model in a turbulent backward-facing step flow.

1Council for Science, Technology and Innovations (CSTI), Cross-ministerial strategic Innovation Promotion program (SIP), Innovative Combustion Technology (Funding agency: JST).

4:18PM A16.00007 Fractional gradient based subgrid-scale models of turbulence1, PATRICIO CLARK DI LEONI, TAMER ZAKI, Johns Hopkins University, GEORGE KARNIAKAKIS, Brown University, CHARLES MENEVEAU, Johns Hopkins University — In large eddy simulations, the effects of the unresolved scales are encapsulated in the turbulence subgrid-scale model. Whether the model can reproduce the correct two-point correlations in the filtered velocity field in LES is governed by its Karman-Howarth equation, and specifically whether the model correctly captures the two-point correlation functions between the stresses and the filtered strain-rates. Inspired by this statistical necessary condition, we develop a model that takes into account non-local effects by using fractional derivatives, and evaluate its performance using data from the Johns Hopkins Database (JHTDB). Starting from direct numerical simulation data of homogeneous isotropic turbulence and channel flows, we filter the data to separate the small and large scales, and calculate the two-point stress-strain rate correlations for the exact case and for models (a-priori) with different fractional orders. We observe that the Smagorinsky model based on standard gradients fails to produce the long range correlations observed in the exact case, while the fractional-gradient models capture the longer tails of the true correlations. As one approaches the wall in channel flow, more complex, highly anisotropic behavior is found.

1The authors thank DARPA for funding this research and NSF for support of the JHTDB

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A17 Energy Harvesting: General 4c4 - Brian Polagye, University of Washington

3:00PM A17.00001 Reliability Study of a Fully-Passive Oscillating-Foil Turbine Concept1, WALTFRID LEE, DYLAN IVerson, University of Victoria, GUY DUMAS, Laval University, PETER OSHKAI, University of Victoria — A self-sustained fully-passive flapping-foil hydrokinetic turbine prototype subjected to water flow at the Reynolds number of 21000 in a water channel. The prototype was exposed to three distinct types of flow disturbances: symmetric vortices shed from an oscillating foil placed upstream of the test foil, boundary layer tripping by distributed roughness on the side face of the foil, and freestream turbulence introduced via a fractal grid turbulence generator. The potential of power extraction of the foil undergoing elastically constrained oscillations in heave and pitch under these nonideal, unsteady flow conditions was quantified by implementing an eddy current brake. When placed in the wake of an upstream oscillating foil, the fully-passive turbine was sensitive to the frequency of the shed vortices in the incoming flow. An overall increase in power extraction was observed when the turbine was subjected to the high freestream turbulence and when the surface roughness was applied.

1Natural Sciences and Engineering Research Council of Canada (NSERC)
3:13PM A17.00002 Vortex Trajectory of High Pitch-High Heave Oscillating Hydrofoils, BERNARDO LUIZ ROCHA RIBEIRO, JENNIFER FRANCK, University of Wisconsin, Madison — Oscillating pitching and heaving foils have been shown to be an efficient mechanism for harvesting hydrokinetic energy. Due to the high pitch and heave amplitudes required for energy production, oscillating foils produce a wake of coherent vortices that interact with downstream foils in array configurations. In order to better predict the vortex-foil interactions, the formation and trajectory of the primary leading edge vortex (LEV) is explored computationally. The hydrofoil kinematics are varied with reduced frequencies of \( \Omega_c/U_c = 0.1 \) and 0.15, with heave amplitudes of 0.5—2c and pitch amplitudes of 65°—85°. The downstream position and timing of the LEV is relatively independent of Reynolds number, but it is strongly influenced by the kinematics. Differences in flapping frequency produce two different LEV trajectory patterns due to changes in the LEV structure during the heave stroke. At the higher frequency the LEV remains attached longer and thus has a lower convective speed in the wake. As the pitch amplitude is increased the LEV size and its trajectory’s vertical distance increase linearly, however as the heave amplitude is increased the maximum vertical distance traveled by the LEV saturates non-linearly.

3:26PM A17.00003 Experimental and Analytical Limitations of Blockage Corrections1, HANNAH ROSS, BRIAN POLAGYE, University of Washington — Flow confinement can significantly impact the performance of wind and water turbines. These effects become appreciable if the blockage ratio, defined as the turbine’s projected area divided by the cross-sectional area of the tunnel or channel, is larger than 5–10%. Due to experimental and computational limitations, studies are often conducted at blockage ratios that exceed this threshold. To estimate performance in unconfined conditions, analytical corrections can be applied to data collected in confined flow. Multiple analytical corrections based on linear momentum theory have been proposed in the archival literature. A previous study by the authors explored the effectiveness of these corrections when applied to experimental data from an axial-flow turbine and a cross-flow turbine. We found that, overall, estimates for unconfined thrust coefficients were more accurate than estimates for unconfined power coefficients. Here, we explore possible causes for this discrepancy in performance. Specifically, we examine the influence that Reynolds dependence and the assumptions that underpin linear momentum theory have on the effectiveness of the power coefficient corrections.

1This work was supported by the National Science Foundation Graduate Research Fellowship Program under Grant No. DGE-1256082 and the U.S. Department of Defense Naval Facilities Engineering Command.

3:39PM A17.00004 Experimental evidence for coupled-mode flutter in a two-meter long parked wind turbine blade, PIETER BOERSMA, BRIDGET BENNER, TODD CURRIER, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — We have conducted a series of experiments on a relatively large-scale wind turbine blade and observed coupled-mode flutter. Theoretical studies predict that the future wind turbine blades are susceptible to coupled-mode flutter. Experimental validation of this prediction is difficult, due to the inherent complications in conducting experimental work on structures that could become unstable due to flutter. We have built a relatively large model of the NREL 5 MW blade from a flexible plastic. The blade was 2 meters long and was comprised of two 0.5 cm-thick shells bonded together such that the torsional natural frequency and the flapwise natural frequencies had a similar ratio to those in the full-scale. This was important, because the theoretical predictions suggest that the third flapwise and the first torsional natural frequencies coalesce into a coupled mode flutter mode. Our experimental results clearly show this coalescence and the resulting modal oscillations in which a combined flapwise and torsional motion is observed.

3:52PM A17.00005 Dynamics and coupling of inertial particles on the wake recovery and flow entrainment of a wind turbine, SARAH SMITH, KRISTIN TRAVIS, Department of Mechanical and Materials Engineering, Portland State University, HENDA DJERIDI, MARTIN OBLIGADO, LEGI - Laboratoire des coulants Géophysiques et Industriels, RAL BAYON CAL, Department of Mechanical and Materials Engineering, Portland State University — Impacting particles such as rain, dust, and other debris can have devastating structural effects on wind turbines, but little is known about the interaction of such debris within turbine wakes. This study aims to characterize the behavior of inertial particles within the axisymmetric turbulent wake of a wind turbine and the resulting effects on wake recovery. Here a model wind turbine is subjected to varied two-phase inflow conditions with wind as the carrier fluid (\( Re_L = 17.7 \times 10^3 - 39.3 \times 10^4 \)) and polydisperse water droplets (averaging 60 micrometers in diameter) at varied concentrations (\( \phi_v = 9.7 \times 10^{-6} - 2.6 \times 10^{-5} \)). Phase doppler interferometry (PDI) and particle image velocimetry (PIV) were employed at multiple downstream locations, centered with respect to turbine hub height. Analysis considers energy and particle size distribution within the wake focusing on particle entrainment, settling velocity, and preferential concentration. Near wake statistics show similarities to those of turbines in single-phase flows, and show persisting velocity deficits at least as far as 9.5 rotor diameters downstream. Complex particle behavior is evident in the near wake region where small particles are captured within tip vortices and central wake regions.

4:05PM A17.00006 Wind Turbine Icing Physics and A Novel Strategy for Wind Turbine Icing Mitigation1, LINYUE GAO, HUI HU, Iowa State University — Wind turbine icing represents the most significant threat to the integrity of wind turbines in cold weather. By leveraging the Icing Research Tunnel available at Iowa State University (ISU-IRT), an experimental study was conducted to elucidate the underlying physics of the important micro-physical processes pertinent to wind turbine phenomena and explore novel anti-/de-icing strategies for wind turbine icing mitigation. A suite of advanced flow diagnostic techniques, which include molecular tagging velocimetry and thermometry (MTV&T), digital image projection (DIP), and infrared (IR) imaging thermometry, were developed and applied to quantify the transient behavior of wind-driven surface water film/ripariot flows, unstable heat transfer and dynamic ice accreting process over the surfaces of wind turbine blade models. A novel, hybrid anti-/de-icing strategy that combines minimized electro-heating at the blade leading edge and an ice-phobic coating to cover the blade surface was developed for wind turbine icing mitigation. In comparison to the conventional strategy to brutally heating the mass blade surface to keep the blade ice free, the hybrid strategy was demonstrated to be able to achieve the same anti-/de-icing performance with substantially less power consumption (i.e., up to 90% power saving).

1The research work is supported by National Science Foundation under award numbers of OISE1826978, CBET- 1916380 and CMMI-1824840, and Iowa Energy Center of the State of Iowa.
4:18PM A17.00007 Reduced-order transport models for energy and the environment
ZHONG ZHENG, Tsinghua-Sichuan Energy Research Institute — We present recent developments on reduced-order modelling and its applications to energy and environmental processes, such as geological CO₂ sequestration and hydraulic fracturing, which is related to the application of energy resource recovery, such as shale gas recovery. In particular, we introduce a series of gravity current models, which describe the spreading and draining dynamics of supercritical CO₂ that is injected into a saline aquifer, for example. In many situations we consider here, self-similar solutions are available to describe some of the interesting dynamics, considering the effects of buoyancy, surface tension, and heterogeneity, for example. We next introduce a series of experimental work that is related to the dynamics of hydraulic fracturing. Model viscous fluids and elastic solids have been employed for the laboratory-scale experiments, and scaling argument has been performed to describe the spatial and temporal evolution of the hydraulic fractures and some of the parametric dependence. We also point out several future research directions and hopefully bring in more interests on this topic.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A18 Turbulence Measurements

3:00PM A18.00001 Resolvent-mode-based Reconstruction of Wall-bounded Turbulent Flows From Non-time-resolved PIV Measurements¹
MENGYING WANG, MAZIAR HEMATI, University of Minnesota, MITUL LUHAR, University of Southern California — Turbulent flows are characterized by broadband spatio-temporal fluctuations, which makes the acquisition of fully-resolved velocity measurements challenging. The goal of this study is to use a physics-based model—projecting the velocity field onto resolvent modes—to reconstruct velocity field from non-time resolved 2D PIV measurements in turbulent channel flow. The resolvent modes are generated via a gain-based decomposition of the governing equations, ensuring physical consistency. A large database of resolvent modes is generated. The Forward Regression with Orthogonal Least Squares algorithm is then used to identify the dominant resolvent modes and to calibrate their amplitude and phase. After calibration, the velocity field can be reconstructed at arbitrary spatiotemporal resolution using the weighted resolvent modes. The weighted resolvent modes also enable estimation of out-of-plane components of velocity and pressure. For proof-of-concept tests of this method, we use DNS data of turbulent channel flow from the Johns Hopkins Turbulence Database. Reconstruction error is quantified and compared with previous studies using Rapid Distortion Theory and Taylor's Hypothesis for reconstruction.

³CVK is grateful for financial support from the graduate school at University of Southern California through the Provost fellowship

3:13PM A18.00002 An eigen-ensemble-variational algorithm for identifying scalar sources from remote measurements in turbulent environments¹
QI WANG, TAMER ZAKI, Johns Hopkins University — The ability to identify the location and intensity of a scalar source in turbulent environment from remote measurements is obfuscated by the stochasticity of turbulent eddies and by diffusion. An algorithm is proposed to solve this inverse problem, which relies on estimating the left and right singular vectors of the scalar impulse-response system, or its eigen-sources and eigen-measurements. The projection of the true source onto an eigen-source is proportional to the projection of the sensor signal onto the corresponding eigen-measurement, and the proportionality is given by the singular value. When only the sensor signal is available, the unknown source is identified by requiring that it accurately reproduces this proportionality. A pre-requisite of the algorithm is knowledge of the eigen-spectrum of the system, which can be available from historical data or approximated using proper orthogonal decomposition of the observation matrix from an ensemble of trial sources. We demonstrate that using only five ensemble members, the source location and intensity are predicted with less than 10% error, and we quantify the effect of sensor noise. Furthermore, the algorithm utilizes forward simulations only and can be easily adopted with expanding time horizon of measurement.

¹This work was funded by the Office of Naval Research (grant N00014-16-1-2542)

3:26PM A18.00003 On the dynamics of Reynolds stresses in the tip region of axial turbomachines¹
HUANG CHEN, The Johns Hopkins University, YUANCHAO LI, DAVID TAN, JOSEPH KATZ, Johns Hopkins University — Experiments examining the flow in the tip regions of axial turbomachines have been performed in a refractive index matched facility, enabling unobstructed optical access. Stereo PIV measurements in closely spaced planes provide high-resolution 3D distributions of the strain, pressure diffusion and velocity–pressure gradient tensor in the Reynolds stress transport equation in canonical turbulent flows is of critical importance for calibrating and improving turbulence models for RANS (Reynolds-Averaged Navier Stokes) based flow simulation. Recent work of Liu and Katz (2018) based on planar-PIV shows the complex nature of the pressure related terms and their substantial impact on the dynamics of turbulence transport throughout a shear layer flow past an open cavity. They also demonstrate the need for a full three-dimensional characterization of the pressure-related turbulence transport terms around the cavity trailing corner. To address this need, time-resolved tomographic PIV measurements of a turbulent shear layer flow past an open cavity is being carried out. The curl-free property of the pressure gradient is used to control the quality of the measured pressure gradient, and the continuity equation is used to control the quality of the velocity measurement. The incoming flow quality is fully characterized so as to facilitate CFD simulation. Detailed velocity and Reynolds stress profiles and turbulence spectrum at selected locations, as well as preliminary tom-PIV data of the flow field just above the cavity trailing corner, will be presented.

¹Sponsored by NASA Glenn Research Center

3:39PM A18.00004 Time-resolved tomographic PIV measurements of a turbulent shear layer flow past an open cavity¹
JOSE MORETO, San Diego State University; University of California, San Diego, XIAOFENG LIU, San Diego State University — Characterization of the pressure-related turbulence terms including pressure–rate-of-strain, pressure diffusion and velocity–pressure-gradient tensor in the Reynolds stress transport equation in canonical turbulent flows is of critical importance for calibrating and improving turbulence models for RANS (Reynolds-Averaged Navier Stokes) based flow simulation. Recent work of Liu and Katz (2018) based on planar-PIV shows the complex nature of the pressure related terms and their substantial impact on the dynamics of turbulence transport throughout a shear layer flow past an open cavity. They also demonstrate the need for a full three-dimensional characterization of the pressure-related turbulence transport terms around the cavity trailing corner. To address this need, time-resolved tomographic PIV measurements of the 3D pressure-related terms in the Reynolds stress transport budget for a turbulent shear layer past a cavity is being carried out. The curl-free property of the pressure gradient is used to control the quality of the measured pressure gradient, and the continuity equation is used to control the quality of the velocity measurement. The incoming flow quality is fully characterized so as to facilitate CFD simulation. Detailed velocity and Reynolds stress profiles and turbulence spectrum at selected locations, as well as preliminary tom-PIV data of the flow field just above the cavity trailing corner, will be presented.

¹Sponsored by ONR
3:52PM A18.00005 Automatic identification and characterization of bursting periods in a turbulent velocity field1, RONI HILEL GOLDSHMIID, DAN LIBERZON, Technion - Israel Institute of Technology, T-SAIL TEAM — A new automatic method for accurate detection of bursting periods in single-point velocity field records is presented. Identification of bursting periods is made by locating an instantaneous increase in the normalized “instantaneous” TKE dissipation rate, obtained using moving window averaging. Use of the record rms and average values for normalization eliminate the need to define flow specific thresholds, and hence decouples burst identification from the generation mechanism. This potentially makes the method universally applicable across various types of turbulent flows. The method performance is examined using a dataset of buoyancy driven turbulent boundary layer flow. Turbulent statistics of the identified non-bursting periods show distinguished similarity to those expected in canonical turbulence, while the identified bursting period statistics differ significantly. To examine the bursting period generation mechanism, statistical findings of temperature fluctuations are examined, and additional tests are provided to assist in identification of the generation mechanism. Examination of the flow field scalar variations in connection with turbulent bursting periods can assist in further understanding of bursting generation and scalar transfer processes.

1The authors gratefully acknowledge the support of this study by the United States-Israel Binational Science Foundation under Grant 2014075.

4:05PM A18.00006 Lagrangian acceleration time scales in anisotropic turbulence1, ROMAIN VOLK, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, Lyon, France, PETER HUCK, NATHANAEL MACHICOANE, University of Washington - Department of Mechanical Engineering, Seattle, WA, USA — We present experimental Lagrangian measurements of tracer particle acceleration auto-correlation functions in an anisotropic and inhomogeneous flow spanning the typical range of experimentally accessible Reynolds numbers. The large scale forcing of the flow creates a stagnation point topology where straininduced motion governs the anisotropic velocity and acceleration fluctuations. We show that the time scales of the acceleration components remain anisotropic even at high Reynolds numbers and that they are related to the dissipative time scale by the Lagrangian structure function scaling constants \( C_0 \) and \( a_0 \). The scaling relation proposed herein is supported by observations using experimental Lagrangian trajectory data sets and analytical calculations using a jointly-Gaussian two-time stochastic model. Examination of acceleration spectra shows that acceleration fluctuations become isotropic in the dissipative range which suggests that the acceleration time scale is not only determined by small scales, but also by large and anisotropic scales whose contributions are substantial, even in the high Reynolds number limit.

1IDEXLYON (ANR-16-IDEX-0005)

4:18PM A18.00007 Inter-scale energy budget in a von Kármán mixing tank, ANNA N. KNUTSEN1, PAWEL BAJ, NICHOLAS A. WORTH, JAMES R. DAWSON, Norwegian University of Science and Technology, JOHN M. LAWSON, University of Southampton, EBERHARD BODENSCCHATZ, Max Planck Institute of Dynamics and Self-Organisation — The inter-scale energy budget in the center region of a von Kármán mixing tank has been investigated based on fully resolved measurement data from volumetric and stereo PIV experiments at \( Re_L = 199 \). The Kármán-Howarth equation generalized for non-isotropic, inhomogeneous turbulence (sometimes referred to as the Kármán-Howarth-Monin-Hill equation) is used to map the full energy transfer, and determine the importance of the different mechanisms transporting energy. The results show that the mean flow, despite its small magnitude relative to the turbulent fluctuations, plays a significant role in the local inter-scale energy transfer. This result stands out from similar investigations of other types of flows, where the mean flow has been found to have a negligible role in transporting energy across scales. This large contribution is caused by the strong gradients in all directions of the mean flow, which is a special feature of the von Kármán flow. The isotropy of the various terms is also evaluated, and strong local variations of the different terms in scale space are observed even down to very small separations, highlighting the importance of using planar or volumetric data when energy transfer is studied.

1Membership Pending

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A19 Advanced Turbulence Models I 401 - Todd Oliver, University of Texas, Austin

3:00PM A19.00001 Physics-informed deep neural networks applied to scalar subgrid flux modeling in a mixed DNS/LES framework, GAVIN PORTWOOD, Los Alamos National Laboratory, MISHA CHERTIKOV, University of Arizona, BALASUBRAMANYA NAIDIGA, JUAN SÀENZ, DANIEL LIVESCU, Los Alamos National Laboratory — The application of artificial neural networks (ANNs) to turbulence closure has been an emergent and active area of research in recent years due to the success of such data-driven methods in fields of computer vision, natural language processing, and other industrial and scientific disciplines. In this research, we apply ANNs to spatio-temporal dynamic modeling of the subgrid passive scalar flux as it relates to large-eddy simulations (LESs). By training on direct numerical simulations (DNSs) of homogeneous isotropic turbulence coupled to a passive scalar, we optimize ANNs to predict the subgrid scalar flux as a function of resolved-scale features. Trained models are then implemented in simulation and evaluated with a-posteriori analysis. In these simulations, filtered scalar advection is coupled to explicitly filtered and statistically-stationary turbulence such that scalar dynamics have no dependence on potentially inaccurate subgrid stress models. By analysis with single- and multi-point statistics, we demonstrate that the data-driven models compete, and often out-perform, properly optimized canonical models. We suggest that this simulation framework may serve as a simplified closure testbed for the investigation and evaluation of data-driven turbulence closures.

3:13PM A19.00002 Fractional physical-inform neural networks (fPINNs) for turbulent flows1, FANGYING SONG, GUOFEI PANG, Division of Applied Mathematics, Brown University, CHARLES MENEVEAU, Mechanical Engineering, Johns Hopkins University, GEORGE KARNIADAKIS, Division of Applied Mathematics, Brown University — We employ fractional operators in conjunction with physics-informed neural networks (PINNs) to discover new governing equations for modeling and simulating the Reynolds stresses in the Reynolds Averaged Navier-Stokes equations (RANS) for wall-bounded turbulent flows at high Reynolds number. In particular, we develop a simple one-dimensional model for fully-developed wall-turbulence that involves a fractional operator with fractional order that varies with the distance from the wall. We use available DNS data bases to infer the function that describes the fractional order, which has an integer value at the wall and decays monotonically to an asymptotic value at the centerline. We show that this function is universal upon re-scaling and hence it can be used to predict the mean velocity profile at all Reynolds numbers. We also extend the fractional RANS for fully-developed turbulent channel flow to a turbulent boundary layer and infer the fractional order in the wake region.

1This work is supported by the DARPA-AIRA grant HR00111990025.
3:26PM A19.00003 Wall-modeled large-eddy simulation of a turbulent channel flow based on artificial neural network1, YOUNG MO LEE, UNIST, JUNGIL LEE, Ajou University, JAE HWA LEE, UNIST – Because the computational cost of large-eddy simulation (LES) in the near-wall region of wall-bounded flows is proportional to approximately square of the friction Reynolds number ($Re_\tau$), utilizing wall-modeled LES (WMLES) is promising to simulate a turbulent flow at sufficiently high Reynolds number with a reasonable cost. The most widely used wall model is an equilibrium stress model (i.e., wall-stress model) based on the momentum conservation. However, this method still needs to improve the accuracy and applicability for complex flows (e.g., swirled or separated flow) due to the limitations of the equilibrium assumption. In the present study, we employ an artificial neural network (ANN) to obtain information of the wall shear stress for WMLES. The proposed method shows good prediction on the mean velocity and Reynolds stress profiles compared to previous models in a turbulent channel flow in the range of the friction Reynolds numbers ($395 < Re_\tau < 5200$), even though the turbulent statistics at untrained Reynolds numbers are predicted.

1This research was supported by the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2017R1D1A1A09000537) and the Ministry of Science, ICT & Future Planning (NRF-2017R1A5A1015311).

3:39PM A19.00004 Zonal Turbulence Modeling via Decision Trees..1, RACHEET MATAI, Geminus.AI, PAUL DURBIN, Iowa State University – The idea of a zonal model is a given model, but with its coefficients varying in different regions of a flow. That idea suggests using a form of classifier to identify zones. The bag-of-trees algorithm has been used to devise a zonal k-omega model. The training data are optimized, coefficient discrepancy fields, obtained by the method of Duraisamy, et al, 2015. The optimization is done with LES data as the target for flow over a circular arc bump. The discrepancy data are binned, with each bin assigned a particular range of values. The zones are parameterized by training the machine learning model with a local feature set. The features are coordinate invariant flow parameters. The classification that is derived by ML is close to associating zones with adverse and favorable pressure gradients. The correction produced by the machine learning algorithm is self-consistent; i.e. once the solution converges, the zones remain fixed.

1Funded by National Science Foundation Grant No. 1507928 and NASA grant NNX15AN98A. We are grateful for Prof. K. Duraisamy for providing data on optimized coefficients.

3:52PM A19.00005 Fractional-Order LES Subgrid-Scale Modeling: Theory and Numerics1, MEHDI SAMIEE, ALI AKHAVAN-SAFAEI, MOHSEN ZAYERNOURI, Michigan State University – Filtering the Navier-Stokes equations in the large-eddy simulation of turbulent flows inherently introduces nonlocal features in the subgrid scale fluid motion. Such long-range effects become even more pronounced when the filter- width enlarges. That urges the development of new nonlocal closure models, which respect the corresponding non-Gaussian statistics of the subgrid stochastic motions. Starting from the filtered Boltzmann equation, we model the corresponding equilibrium distribution function with a Lévy stable distribution, which leads to the proposed fractional-order modeling of subgrid-scale stresses. We approximate the filtered equilibrium distribution function with a power-law term, and we derive the corresponding fractional Laplacian, $(Δ)^α$, $α ∈ (0, 1]$, of the filtered velocity field. Finally, the introduced model undergoes a priori evaluations based on the direct numerical simulation database of forced and decaying homogeneous isotropic turbulent flows at high and moderate Reynolds numbers, respectively.

1This work was supported by the AFOSR Young Investigator Program Award (FA9550-17-1-0150), and partially by MURI/ARO (W911NF-15-1-0562), and by ARO Young Investigator Program Award (W911NF-19-1-0444).

4:05PM A19.00006 Motivation and development of the time-averaged active model-splitting hybrid RANS/LES1, SIGFRIED HAERING, Argonne National Laboratory, TODD OLIVER, Oden Institute for Computational Engineering and Sciences, RAMESH BALAKRISHNAN, Argonne National Laboratory, ROBERT MOSER, Oden Institute for Computational Engineering and Sciences — We discuss motivation and development of the time-averaged active model-split (TAMS) hybrid RANS/LES approach. TAMS has been specifically constructed to overcome challenges associated with existing hybrid approaches related to LES/RANS blending techniques and inconsistencies between the resolved and modeled turbulence. Core to TAMS is a hybridization strategy in which the RANS and LES components act through separate models formulated using the mean and fluctuating velocity, respectively, as approximated by time averaging over the local turbulent timescales. Justification for this splitting strategy is discussed based on true subgrid terms from filtered DNS and simple LEVM arguments. Multiple validation cases are used to demonstrate the potential of the method.

1This work is supported by NASA “RESEARCH OPPORTUNITIES IN AERONAUTICS 2015” and used resources of the ALCF, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357

4:18PM A19.00007 Numerically measured scale-dependent eddy viscosity in homogeneous isotropic turbulence, YASAMAN SHIRIAN, ALI MANI, Stanford University — In the present work, we report the directly measured eddy viscosity in incompressible homogeneous isotropic turbulence using the macroscopic forcing method. Our results provide a scale-dependent eddy viscosity; specifically, in the low-wavenumber-limit, eddy viscosity is a constant, while in the high-wavenumber limit, it becomes inversely proportional to the wavenumber. Our results present a contrast to the previously reported eddy diffusivity via renormalization group theories (Yakhot and Smith 1992), this difference is attributed to proper quantification of transport at small scales by large-eddies in the present study. We also report the scale-dependent turbulent Schmidt number as well as collapse of the measured eddy-viscosity operator with increase of flow Reynolds number. Supported by DOE

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A20 Geophysical Fluid Dynamics General I 602 - Jerome Neufeld, Cambridge
3:00PM A20.00001 Formulation and implementation of free-slip boundary condition on a deformed domain in the context of continuous Galerkin high-order element-based discretization. 1, THEODOROS DIAMANTOPOULOS, PETER DIAMESSIS, Cornell University, MAREK STASTNA, University of Waterloo — The free-slip condition is a convenient choice of boundary condition (BC) used in the simulation of stratified flows to circumvent the resolution of fine-scale no-slip-driven bottom boundary layers when numerically solving the incompressible Navier-Stokes equations (INS). The nodal spectral element method is a commonly used strategy in the discretization of the INS where the free-slip condition arises as a natural BC from the integration-by-parts of the viscous term in the INS. In the context of a deformed domain, the tangential component of the force associated with the free-slip condition, couples the Cartesian velocity components leading to a larger system of equations for the calculation of the velocity field. One approach to mitigate this complexity is to impose a pseudo-traction BC, where each velocity component can be solved for independently. Effectively, the velocity components can be treated as scalar quantities allowing the use of the same computational machinery as for the calculation of the density and the pressure. In a series of test problems, which include the propagation of an internal solitary wave, the effective numerical drag produced by the pseudo-traction BC will be quantified thereby enabling the assessment of the accuracy of the pseudo-traction approach.

1 NSF-OCE 1634257

3:13PM A20.00002 Role of aeolian hysteresis and secondary turbulence in dust entrainment over arid landscapes, SANTOSH RANA, WILLIAM ANDERSON, UT Dallas — Saltation, the wind-blown hopping motion of sand particles, plays an effective role in dust eruption from the sediment bed. Low and high momentum regions appear as long streaks in atmospheric turbulence. Positive and negative vertical velocity are associated with low and high momentum regions respectively. High momentum regions initiate saltation when the fluid velocity exceeds the fluid threshold value and saltation continues till the fluid velocity drops below the impact threshold value. This phenomenon is called hysteresis or the lag between the initiation and cessation of saltation. Saltation occurs during this hysteresis period. The sweeping motion and the negative vertical velocity of the high momentum regions do not allow entrainment of the dust released during saltation. However, the positive vertical velocity associated with the low momentum region picks up the dust. A conditional averaging method is employed to study this paradox as two modes of dust entrainment. In the primary mode, the dust released by the saltating particles of the high momentum region is entrained by the neighboring low momentum region. In the secondary mode, dust released due to saltation by a very recent high momentum region is entrained by a closely following low momentum region in the flow direction. Here, we explain how dust exposed to the low momentum region as a result of hysteresis gets entrained by two modes of transport. The methodology explained here is applicable to both earth and mars conditions.

3:26PM A20.00003 Revisiting eddy diffusivity models in geophysical boundary layers, TOMAS CHOR, JAMES MCMILLIANS, MARCELO CHAMECKI, University of California, Los Angeles — Eddy diffusivity models have proved invaluable when modeling turbulent fluxes in many situations. However, in atmospheric and oceanic boundary layers, large-scale motions (for example convective plumes and Langmuir circulations) do not conform to the usual assumptions necessary for their application. This has prompted the creation of several ad-hoc models, each designed to work under rather specific conditions, and each of them often failing to work well outside their intended operating area. In this work we present an alternative unifying solution that estimates the total eddy diffusivity without a priori assumptions about its shape or scaling. The approach is based on the fact that the eddy diffusivity should depend only on the flow, which we use as a basis for an optimization procedure that uses Large-Eddy simulation data. The result of our approach is that most of the fluxes are modelled with an eddy diffusivity, while the rest (which depends on specific sources of scalars and is attributed to large scale motions) is modelled as non-dissipative processes. We present an approach of our proposed model to the classic case of a convective boundary layer and show that we are able to predict the heat flux from quantities measured using passive tracers.

3:39PM A20.00004 Semi-Empirical Models of Surface Pressure in the Planetary Boundary Layer in terms of Second-Order Structure of Atmospheric Turbulence, GREGORY LYONS, U.S. Army Engineer Research And Development Center, Construction Engineering Research Laboratory, CARL HART, U.S. Army Engineer Research And Development Center, Cold Regions Research and Engineering Laboratory — The fluctuations in surface pressure beneath turbulent boundary layers have long been of interest due to the aerodynamic noise and structural vibrations they induce. In the planetary boundary layer, analogous fluctuations at the ground surface contribute to instrument noise directly as static pressure, recorded by micrometers and microphones, and indirectly as ground motion, detected by seismographs and geophones. It is hypothesized that the fast (i.e. linear) term of the pressure Poisson equation make the principal contribution to surface fluctuations, so that only second-order structure of the inhomogeneous velocity field is necessary for modeling of second-order pressure statistics. The mirror flow model due to Kraichnan and rapid-distortion theory models proposed by Mann, which both derive inhomogeneous turbulence from initially homogeneous fields, are used to produce semi-empirical models of the surface pressure wavenumber spectrum. With an effective convection velocity, frequency pressure spectra are derived and contrasted. Using meteorological observations from a recent experiment, model parameters are estimated from velocity spectra, and the resultant pressure spectral models are compared with measurements from flush-mounted pressure sensors.

3:52PM A20.00005 The poroelastic aquifer: river flux response to solid Earth tidal forcing, JEROME NEUFELD, University of Cambridge, ERIC LAJEUNESSE, OLIVIER DEVAUCHELLE, Institut de Physique du Globe de Paris — The water flux through rivers is chieflly determined by the intermittent charging of groundwater in laterally extensive shallow aquifers by rainfall and drawdown by seepage to the river bank. Here we show that the pattern of drawdown is modulated by the small-scale forcing of shallow poroelastic aquifers by the solid Earth tides. We use a shallow poroelastic model of flow in a deformable matrix to show that the modulation of the solid stress by solid Earth tides gives rise to the observed 2-4 cm height variation in the far-field aquifer depth. This oscillatory groundwater table also drives a modulation of the water flux into the river which is apparent in long-term records of stream flux. By understanding the poroelastic response of aquifers to solid Earth tides as observed in the river flow data, we are therefore able to infer properties of distributed aquifers from readily available records of the river flow. The fluid dynamical modelling, motivated by observations at the Quick creek catchment, Guadeloupe, also demonstrates a generic framework for understanding the interaction between periodic solid stresses and fluid flow in deformable porous media.
University the interior after approximately 200 Ma, which may explain the long persistence of the Lunar dynamo on temperature, pressure and solid fraction. The resulting surface heat flux depends primarily on the temperature of the mixed interior and on explain the long timescales for crust formation. This involves convection dominated by a temperature-dependent viscosity, producing a stagnant-lid, may be more relevant to the early lunar magma ocean and explain the long timescales for crust formation. This involves convection with in a fluid whose viscosity varies by several orders of magnitude as a function of temperature and crystal fraction. The temperature scale most relevant to the convection is that over which the viscosity varies. This scale is greatly reduced during rapid changes in viscosity, such as when jamming occurs at a critical melt fraction. We consider a one-dimensional model in which the lunar magma ocean is considered as one body of convecting silicate, with a viscosity that depends strongly on temperature and solid fraction. The resulting surface heat flux depends primarily on the temperature of the mixed interior and on the hydrostatic pressure at the base of the stagnant lid. This results in an initial unstable scenario with a lid that is mostly entrained back into the interior after approximately 200 Ma, which may explain the long persistence of the Lunar dynamo.

Saturday, November 23, 2019 3:00PM - 4:31PM –
Session A21 Drops: Temperature and Marangoni Effects I

3:00PM A21.00001 Freezing-damped drop impacts , CALLUM WATSON, JEROME NEUFELD, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, CHLO MICHAUT, Laboratoire de Gologie, Terre, Plante, Environnement, cole Normale Superieure de Lyon, France — Typical models of the thermal evolution of the Moon involve a solid crust growing via crystal flotation, with an underlying magma ocean then cooling by conduction through the crust. However, convection dominated by a temperature-dependent viscosity, producing a stagnant-lid, may be more relevant to the early lunar magma ocean and explain the long timescales for crust formation. This involves convection with in a fluid whose viscosity varies by several orders of magnitude as a function of temperature and crystal fraction. The temperature scale most relevant to the convection is that over which the viscosity varies. This scale is greatly reduced during rapid changes in viscosity, such as when jamming occurs at a critical melt fraction. We consider a one-dimensional model in which the lunar magma ocean is considered as one body of convecting silicate, with a viscosity that depends strongly on temperature and solid fraction. The resulting surface heat flux depends primarily on the temperature of the mixed interior and on the hydrostatic pressure at the base of the stagnant lid. This results in an initial unstable scenario with a lid that is mostly entrained back into the interior after approximately 200 Ma, which may explain the long persistence of the Lunar dynamo.

3:13PM A21.00002 Dynamics and temperature evolution during droplet impacts on cold surface , FENG WANG, MAN HU, DAOSHENG DENG, Fudan University, Shanghai, China — Droplet impacting on cold surface is significant for many practical situations in our daily life, such as anti-icing on wind turbines, aircraft and power transmission lines. By using high speed camera and thermal camera, here we report the dynamics and temperature evolution during water droplet impacts and freezes on cold metal surface. In the impact process, the maximum spread radius is found to have a 1/3 power law with the initial height, which is different from the impacting behavior at the room temperature. In the retraction process, three different morphology are indentified when changing the initial height from low to high, which is puddle, transition state and pancake. In the freezing process, the ice-water boundary and the zero degree isotherm are extracted from the high speed videos and thermal profiles. The ice-water boundary retraction speed is found to be slower than the zero degree isotherm, which indicates the subcooling freezing process.

3:26PM A21.00003 Arrested dynamics of droplets impacting icy surfaces , FARZAD AHMADI, ANDREW FUGARO, Virginia Tech, SAURABH NATH, Laboratory PMMH, ESPCI Paris, PSL Research University, France, LadHyX, Ecole polytechnique, Palaiseau, France, HYUNG-GON PARK, JONATHAN BOREYKO, Virginia Tech — We study the competition between the spreading and freezing dynamics of droplets impacting icy surfaces. Experiments were conducted on two different frosted surface configurations: planar or suspended cable. The dynamics of droplet impact were captured using a side-view high-speed camera, where the droplet was initially either at room temperature or close to the freezing temperature. For droplets spreading on the planar substrate, the advancing contact line was arrested significantly faster with decreasing surface temperature or initial droplet temperature. Droplets impacting the icy cable either detached and fell from the cable or were captured and frozen, depending upon the Weber number, surface temperature, droplet temperature, and the ratio of the droplet and cable diameters. A scaling model elucidated that the extent of droplet spreading is a balance between capillary-inertial effects and the specific cooling of the droplet toward its freezing temperature.

3:39PM A21.00004 Coalescence dynamics of a droplet on a heated pool , PANKAJ KOLHE, PAVAN KUMAR KIRAR, Indian Institute of Technology Hyderabad, India, KATHRYN ALVARENGA, Texas A&M University, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We experimentally investigated the coalescence dynamics of an ethanol drop in ethanol pool maintained at a higher temperature than the drop. The size of the impacting drop and the temperature difference between the pool and the drop are varied and their effect on the secondary droplet formation has been studied. It is observed that increasing the temperature of the liquid pool, has a non-monotonic effect on the partial coalescence phenomenon. Increasing the droplet size increases the size of the secondary drop for all values of the temperature differences considered in the present study.
3:52PM A21.00005 Evaporation of ethanol-water sessile droplet of different compositions at an elevated substrate temperature. SARAVANAN BALUSAMY, SAYAK BANERJEE, PRADEEP GURRALA, PALLAVI KATRE, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We experimentally investigate the evaporation dynamics of sessile droplets with different compositions of ethanol-water binary mixture at different substrate temperatures. At elevated substrate temperature, we observed an early spreading stage, an intermediate pinned stage and a late receding stage of evaporation. Increasing the substrate temperature decreases the lifetime of binary droplets rapidly. We found that the lifetime of the droplet exhibits a non-monotonic trend with increase in ethanol concentration in the binary mixture, which can be attributed to the non-ideal behaviour of water-ethanol binary mixtures. Interestingly, the evaporation dynamics for different compositions at high substrate temperature exhibits a self-similar trend showing a constant normalised volumetric evaporation rate for the entire evaporation process. This indicates that the evaporation dynamics of a binary droplet of a given composition at high substrate temperature is equivalent to that of another pure fluid with a higher volatility at room temperature. The evaporation rates of pure and binary droplets at different substrate temperatures are compared against a theoretical model developed for pure and binary mixture droplets. The model predictions are found to be quite satisfactory.

4:05PM A21.00006 Evaporation of ethanol-water sessile droplet on an inclined substrate at an elevated temperature. SAYAK BANERJEE, SARAVANAN BALUSAMY, PALLAVI KATRE, PRADEEP GURRALA, KIRTI SAHU, Indian Institute of Technology Hyderabad, India, PHASE CHANGE IITH COLLABORATION — We experimentally investigated the evaporation dynamics of water-ethanol binary droplets of various molar compositions on an inclined heated substrate. The substrate temperature and inclination angle are varied and the droplet behaviour are observed using a combined optical and thermal imaging techniques in a custom-made goniometer-IR camera set-up. The triple line behaviour, the contact angle and the wetting diameter evolution of the droplet, as well as thermo-solutal Marangoni wave dynamics are studied. Theoretical modelling of evaporation rates for binary-droplets on heated inclined substrates has been performed and compared against the experimental results.

4:18PM A21.00007 Residues formation on surface of heated wire after drop impact: correlation between solution drop impact dynamics and residues thickness. KYEONGMIN KIM, SANG JUN LEE, WONJONG CHOI, Korea University — Impact between solid structures and solutions essentially leaves residues on surfaces, such as calcite in water. While the impacting angle and velocity of the solution affect the formation of the residues, the temperature regime of the solid surface is another significant factor because it intrinsically manipulates the contact interfaces between liquids and solids, as well as their solubility. Herein, we report a systematic study of the residual nanomaterials after a drop impact containing carbon nanotubes (CNTs) on a heated metal wire in various temperature range. While a drop casting method for the individual droplet comprising CNTs induces the impact with 2-mm nichrome wires in diameter, a joule-heating method controls the wire temperature in 25-400 C. The phase diagram in terms of the impact velocity, wire temperature, and formation of the residues are quantitatively investigated to elucidate the correlation between the solution drop impact dynamics and the resulting CNT residue thicknesses. The fundamental understanding of the residual nanomaterials, induced by the impact between solid structures and solutions in wide temperature ranges would provide the insight for a robust design of broad engineering systems such as heat exchangers, solar power modules, and turbines.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A22 Drops: Coalescence I 604 - Sumeet Thete, Air Products

3:00PM A22.00001 Internal Flows in Impacting and Coalescing Droplets of Different Surface Tension. THOMAS C. SYKES, University of Leeds, ALFONSO A. CASTREJON-PITA, University of Oxford, J. RAFAEL CASTREJON-PITA, Queen Mary University of London, MARK C. T. WILSON, DAVID HARBOTTLE, ZINEDINE KHATIR, HARVEY M. THOMPSON, University of Leeds — Internal flows in impacting and coalescing mm-sized droplets, with different properties, in contact with a substrate is studied using two colour high-speed cameras (side and bottom views). This arrangement allows internal and surface phenomena to be distinguished, and hence the true extent and mechanism of mixing to be determined. Given enough lateral droplet separation, the impact droplet inertia generates a surfactant wave, which can be enhanced or suppressed through Marangoni flow by modifying the surface tension difference between the droplets. Conditions promoting good mixing are established, with practical implications for lateral separation and deposition order.

3:13PM A22.00002 Numerical investigation of collision of two liquid metal droplets under the influence of magnetic field. JIAICAI HUANG, JIE ZHANG, Xi’an Jiaotong University China, KIRTI SAHU, Indian Institute of Technology Hyderabad, India, MINGJIU NI, University of Chinese Academy of Sciences, Beijing, China — We investigate the collision dynamics between liquid GaInSn droplets under the influence of an applied magnetic field by conducting three-dimensional numerical simulations. The surface tension between liquid metal droplets is much larger than that of the normal liquid droplets. The collision dynamics is influenced by a large number of parameters, namely, the Weber number, Reynolds number, separation distance and the relative velocity of the droplets, along with the applied magnetic field. We have validated our solver by comparing with the earlier experimental results without the magnetohydrodynamics (MHD) effect. Different regimes of coalescence and separation dynamics are observed without the MHD effect and found to show excellent agreement with the earlier experiments. Then we study the effect of the external magnetic field and its direction on the droplet dynamics and regime map of the distinct coalescence behaviors. The underlying physics of the collision dynamics of liquid metal droplets has been analyzed using the resultant flow field.

3:26PM A22.00003 Coalescence dynamics of a droplet on a sessile droplet. MANISH KUMAR, RAJNEESH BHARDWAJ, Department of Mechanical Engineering, Indian Institute of Technology Bombay, Mumbai 400076, India, KIRTI CHANDRA SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Sangareddy 502 285, Telangana, India — The coalescence dynamics of an ethanol droplet freely falling on a sessile ethanol droplet resting on glass substrate is investigated experimentally using a high-speed imaging system. The sessile glass surface is found to be 24with a standard deviation of 1.3. The regime maps showing the partial and the complete coalescence behaviors in the plane of the normalized impact height and the volume of the sessile droplet with the diameter and the volume of the impacting droplet are presented. It is observed that the partial coalescence of satellite droplet occurs when the ratio of the volume of the sessile droplet to that of the impacting droplet is greater than two. The size of the satellite droplet is found to be about 0.1 times the size of that of the impacting droplet, which increases with the increase in the normalized impact height and normalized volume of the sessile droplet.
3:39PM A22.00004 Coalescence of Liquid Metal Droplets with application to metallic 3D printing, RYAN MCGUIN, PIROUZ KAVEHPOUR, ROB CANDLER, UCLA — In the field of additive manufacturing, printing functional metallic parts remains the ultimate challenge and goal of research and industry. Metals present many challenges due to high thermal conductivity, their tendency to oxidize and sensitivity to thermocapillary effects. Understanding and mitigating these variables is necessary for the process to become workable, scalable and economic. For droplet-based 3D printing, it is essential to study the liquid droplet coalescence on a planar surface presents a challenging crucible of physical forces including viscosity, capillarity, inertial and gravitational body force. This is further complicated in some of applications such as droplet coalescence in a “quasi-film” confined to the air-metal or liquid-metal (in the case of surrounding the droplet with an immiscible liquid) interface. This oxidation layer introduces inhomogeneity of material properties at the surface, as well as interfacial phenomena that defy traditional models of surface tension or interfacial energy driven interactions. We investigated the coalescence of room temperature liquid metal and its unique features. The coalescence process is studied for metallic droplets with and without oxidation. We observed that many features of this phenomenon differ from non-metallic drop coalescence.

3:52PM A22.00005 Droplet propulsion on a superhydrophilic wire induced by coalescence, ALLISON O’DONNELL, YOUMHU JIANG, KYOO-CHUL PARK, Northwestern University — Prior research has reported that droplets can jump from both superhydrophobic flat surfaces and hydrophobic cylindrical wires upon coalescence. As the surface area of droplets reduces after coalescence, released surface energy transitions to kinetic energy, causing the droplet to jump. Our findings indicate that droplet propulsion induced by the same surface-to-kinetic energy transition also occurs on superhydrophilic wires. In this experiment, droplets are sequentially deposited on an inclined superhydrophilic wire at a controlled rate. As the droplets wet the wire and flow downward, adjacent droplets may merge. In such cases, droplet speeds before and after coalescence are measured. This is repeated with droplets of differing viscosities and surface tensions and on wires of varying diameters. The results suggest that the ratio of increased kinetic energy to available surface energy is less than 4%. Additionally, it decreases as viscosity increases, surface tension decreases, and wire diameter increases. This trend can be attributed to the energy loss from viscous friction as droplets oscillate during coalescence. A scaling law is also provided to explain this trend. The phenomena reported in this study provide new insights into liquid transport on wires.

4:05PM A22.00006 Self-Similar Coalescence of Liquid Lenses, MICHIEL HACK, WALTER TESUY, KIRSTEN HARTH, University of Twente, QINGGUANG XIE, JENS HARTING, Eindhoven University of Technology, JACCO SNOEIJER, University of Twente — The coalescence of droplets is of key interest to many industrial applications, such as inkjet printing and spray formation. Here, we study the initial stages of coalescence of liquid lenses, consisting of droplets floating on a liquid pool. Using high-speed imaging, we find that the bridge grows with self-similar dynamics, with different scaling laws at low and high lens viscosities, indicating the existence of two coalescence regimes. We provide an analytical description based on the slender geometry of the system, using the two-dimensional thin sheet equations. Excellent agreement is found with the experiments, capturing both the exponents and the detailed spatial structure of the similarity solution in both regimes. Finally, we show that all data collapse on a single curve capturing the full range of the coalescence dynamics.

4:18PM A22.00007 Shear-induced non-coalescence for contactless droplet bearings, MICHELA GERI, BAVAND KESHAVARZ, GARETH MCKINLEY, Mechanical Engineering, MIT — Coalescence between a drop and a bath of the same liquid can be delayed or even completely suppressed if the lubricating air layer separating the approaching liquid surfaces is replenished. Thermal Marangoni stresses can promote such scenario if an initial temperature difference is imposed between drop and bath. Even when the droplet is left to thermally equilibrate coalescence can be delayed for many seconds, with the delay time being an increasing power law of the temperature difference. Under isothermal conditions coalescence can be suppressed if the droplet is externally forced to move over the liquid surface. This motion continuously replenishes the air cushion separating the droplet and bath while inducing fluid recirculation within both liquids. We use a custom setup that allows us to carefully control the speed at which the drop is kept in motion over the bath. By pinning the droplet to a force transducer we measure the friction force that is generated by the lubricating air flow for different velocities and different fluid viscosities. We discuss our results in view of a lubrication analysis for the air flow within the gap and investigate the possibility of using this technique to generate contactless droplet bearings.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A23 Bubbles: Surfactants and Foams

3:00PM A23.00001 Energy balance and the velocity gap in foam fracture, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, PETER STEWART, School of Mathematics and Statistics, University of Glasgow — It has been shown experimentally and in simulations that a quasi-two-dimensional foam bubble layer, when driven under external stress, can display ruptures of successive films that constitute an analogue to brittle type-I fracture. Simplified discrete and continuum models from first principles of fluid dynamics capture the features of propagation, including the existence of a critical speed below which steady fracture cannot be sustained (velocity gap). In order to understand the magnitude and parameter dependence of the velocity gap, it is advantageous to strike an energy balance in the system and thus connect with continuum dynamical fracture mechanics. The crack dynamics is characterized by system size (because of changes in strain energy concentration) as well as by dissipation effects in the foam. Access to an explicit analytical formalism for the modeling of all physical effects complements the simulation results and reveals general principles governing the occurrence of fracture velocity gaps.

3:13PM A23.00002 Active Foam: Connecting Structure, Dynamics and Control, LAUREL KROOK, Department of Mechanical Engineering, Stanford University, MATTHEW BULL, Department of Applied Physics, Stanford University, MANU PRAKASH, Department of Bioengineering, Stanford University — By inflating and deflating voxels within a polydisperse 2-D air-liquid foam, we demonstrate an experimental system where we perturb soft materials in a radially-symmetric manner. These cyclic perturbations can be coordinated spatially and temporally to encode (“write”) mechanical properties into the material. Using both a short-range mechanism (cascades of neighbor-swapping events) and a long-range mechanism (low-frequency acoustic), we discuss how to achieve a robust and morphable active material. Topics of interest include scaling analyses, noise, and accounting for complexity that arises from polydispersity and initial conditions. The goal of this work is to understand fundamental principles of confluent tissues and develop synthetic analogs.

1This research received funding from NSF (GRFP, 1453190) and CZ Biohub.

1This research was co-financed by Oc-Technologies B.V., University of Twente, and Eindhoven University of Technology.
3:26PM A23.00003 Microfluidic-based foams: a possible template for photonic structures\textsuperscript{1}, ILHAM MAIMOUNI, MARIA RUSSO, MARYAM MORVARIDI, JANINE COSSY, PATRICK TABELING, Ecole Supérieure de Physique et de Chimie Industrielles, PSL, France — Through the past decades, foams have taken more and more place into our modern world and have been used in a myriad of applications such as insulation building materials, food industry and photo-catalysis thanks to their interesting structural properties. Recently, 2D foams have been investigated to be self-assembled materials exhibiting interesting photonic properties. In the present study, we aim at exploring the 3D foams case. In this perspective, microfluidic technologies are used to develop 3D, solid, highly monodisperse polymeric foams by packing air bubbles in aqueous solution containing a polymer. The bubbles are produced in a PDMS (Polydimethylsiloxane) microfluidic chip and directly assembled in a microfluidic channel giving birth to highly tunable 3D foam. Indeed, by varying fluid pressures, the foam composition and the polymerization process, we manage to sharply control bubbles production and thereby govern the structural properties of the obtained material: porosity, pores size, connectivity and polydispersity. Electromagnetic simulations are then performed to study wave propagation in such foams, revealing very interesting transmission regimes and opening the way for a new technological application of fluid-based systems.

\textsuperscript{1}The Microflusa project receives funding from the European Union Horizon 2020 research and innovation programme under Grant Agreement No. 664823.

3:39PM A23.00004 Evaporation induced spontaneous cyclic dimpling in binary mixtures and its role in bubble stability. VINEETH CHANDRAN SUJA, JAVIER TAJUELO, GERALD FULLER, Stanford University — Liquid mixtures having components with heterogeneous volatilities, such as lubricants, are known to support stable foams in the absence of surfactants. Recent studies have shown this is a consequence of stabilizing Marangoni flows induced by the differential evaporation of liquid components from the surface of the bubble. Interestingly, unlike surfactant induced Marangoni flows, evaporation induced Marangoni flows in fact change the thickness of bubble walls resulting in a fascinating phenomenon known as ‘spontaneous cyclic dimpling’. This phenomenon has been studied in the context of emulsion stability resulting from surfactant diffusion, however very little is known about evaporation induced spontaneous cyclic dimpling. Here we study this phenomenon in detail by visualizing the spatio-temporal evolution of the wall thickness of single bubbles in binary mixtures using white light interferometry. Various binary mixtures of silicone oils are tested to understand the mechanics of spontaneous cyclic dimpling and its influence on bubble stability as a function of species mole fraction, volatilities and viscosities.

3:52PM A23.00005 Effects of soluble surfactant and viscoelasticity on pressure-driven turbulent bubbly channel flow\textsuperscript{1}, ZAHEER AHMED, Koc University, Turkey, DAULET IZBASSAROV, PEDRO COSTA, OUTI TAMMISOLA, KTH Mechanics, Sweden, METIN MURADOGLU, Koc University, Turkey — Particle-resolved direct numerical simulations are performed to investigate the combined effects of soluble surfactant and viscoelasticity on structure of pressure-driven turbulent bubbly channel flow ($Re_c = 180$). Incompressible flow equations are solved fully coupled with FENE-P viscoelastic model and soluble interfacial and bulk surfactant concentration equations. A non-linear equation of state relates surface tension to interfacial surfactant concentration. The method is first validated using benchmark turbulent single- and two-phase flows. Then massively parallel simulations are performed to examine effects of viscoelasticity and surfactant on turbulent bubbly flows. We found that clean bubbles move toward the walls due to inertial lift force, resulting in formation of wall-layers and a significant decrease in the flow rate. An addition of strong enough surfactant alters the direction of lateral migration of bubbles resulting in a nearly uniform bubble distribution across the channel. For the viscoelastic case, shear-thinning effects promote inertial lift, enhancing formation of bubbly wall-layers and consequently strong decrease in the flow rate. Formation of wall-layers is determined by the interplay of viscoelasticity and surfactant, when they act together.

\textsuperscript{1}Turkish Academy of Sciences, TUBITAK grant No. 115M688 and the Swedish Research Council through grants No. VR2013-5789 and No. VR 2014- 5001

4:05PM A23.00006 Direct Numerical Simulations of the Three Dimensional Dynamics of Surfactant Laden Bursting Bubbles\textsuperscript{1}, DAMIR JURIC, LIMSI, CNRS, France, RICARDO CONSTATE-AMORES, ASSEN BATCHVAROV, LYES KAHOUADJI, Imperial College London, SEUNGWON SHIN, Hongik University, South Korea, JALEL LAHOUADJI, CRISTIAN CONSTATE-AMORES, RICHARD CRASTER, OMAR MATAR, Imperial College London — Bursting bubbles play an important role in both industrial applications and nature with $\sim 10^{14}-10^{20}$ bubble bursts per second over the oceans, exchanging chemical components or heat between the ocean and the atmosphere. When a bubble is close to a free surface, it forms a hole which leaves an open unstable cavity that will collapse; the change of the interface curvature leads to the formation of a central jet, which breaks into droplets according to the Plateau-Rayleigh instability. The surfactant-free interfacial dynamics are well understood but the surfactant-laden ones are not. Neglecting gravity, the Laplace number is the only remaining control parameter measuring the relative importance of surface tension to viscous forces i.e. $La = \rho R \sigma / \mu^2$, where $\rho$, $\mu$, $\sigma$, and $R$ are the liquid density, viscosity, surface tension, and the initial radius of the droplet, respectively. 3D DNS simulations varying the Peclet number, $Pe = UR/D$, where $U$ and $D$ denote the velocity and diffusion coefficient, respectively, were performed to analyse the fate of the jet. Results regarding the interfacial surfactant concentration distribution, the surface tension gradients, and the importance of Marangoni stresses on the jet formation will be presented.

\textsuperscript{1}Funding from BP, EPSRC, PETRONAS, RAEng.

4:18PM A23.00007 The effect of surfactant on the tail dynamics of elongated bubbles\textsuperscript{1}, OMAR MATAR, ASSEN BATCHVAROV, Imperial College London, MIRCO MAGNINI, University of Nottingham, LYES KAHOUADJI, CRISTIAN CONSTATE-AMORES, RICHARD CRASTER, Imperial College London — Following the classical work by Bretherton and Taylor, the propagation of elongated gas bubbles in micro-channels has received a great deal of attention in the literature due to its relevance to a wide range of applications. Recent work by Magnini et al. Phys. Rev. Fluids, 023601 (2018) and Moran et al. (Bulletin of the American Physical Society, vol. 63, D09.00002) has also examined these flows (with concurrent liquid flow) with significant buoyancy and inertial effects in both vertical and horizontal configurations. The present work focuses on bubble tail undulations in the presence of surfactants whose effects merit attention. We carry out three-dimensional direct numerical simulations of the flow using a hybrid front-tracking/level-set method (Shin et al., J. Comput. Phys. 359, 409-435, 2018) over a wide range of surfactant properties including diffusivity, elasticity, and solubility. The effect of these properties on the speed of bubble propagation and interfacial shape is examined, paying particular attention to the bubble tail dynamics at high Reynolds number.

\textsuperscript{1}We acknowledge gratefully the contribution of Drs Damir Juric and Jalel Chergui (both from LIMSI, CNRS, France), and Dr Seungwon Shin (Hongik University, South Korea), in terms of code development, and funding from EPSRC (grant EP/K003976/1), and the Royal Academy of Engineering (Research Chair for OKM).
3:00PM A24.00001 Linear Stability of Taylor Bubbles in Downward Flowing Liquids\(^1\), HABIB ABUBAKAR, OMAR MATAR, Imperial College London — A Taylor bubble rising against a downward flowing liquid in vertical pipes is known to lose its symmetric shape when the velocity of the liquid exceeds a critical value. The influence of the liquid flow conditions, characterised by the dimensionless Eötvös number, \(E\_o\), and inverse viscosity number, \(N\_v\), on the onset of transition from symmetric to asymmetric bubble shape, is examined using linear stability analysis. To gain insight into the underlying mechanism, an ‘energy’ budget analysis is carried out to isolate the most dominant energy term that drives the instability. The analysis shows that the driving mechanism is dependent on whether or not the effect of surface tension can be neglected. For negligible surface tension effects, the instability originates from within the bubble and the dominant source of energy that drives the instability is related to perturbations in the bubble pressure. In the case of strong surface tension, the mechanism is related to disturbances connected to the interfacial stress condition.

\(^1\)Petroleum Technology Development Fund Scholarship for HAA; RAEng/PETRONAS Research Chair for OKM.

3:13PM A24.00002 Numerical Modelling of Phase-change: Application to Evaporating Taylor Bubbles in Superheated Rectangular Microchannels\(^1\), YAN DELAURE, Dublin City University, THOMAS ABADIE, OMAR MATAR, Imperial College London — In micro-structured devices, the high surface to volume ratio leads to enhanced transport phenomena and the design of micro heat-exchangers requires some fundamental understanding on the dynamics and transfer phenomena taking place. The present study therefore aims at presenting a simple but accurate continuous model for modelling phase change in microfluidic devices within a finite volume Navier-Stokes flow solver. The proposed model is based on a Level Set method for capturing the interface on a fixed structured cartesian mesh and a continuous surface force method for the capillary force. The temperatures are solved in each phase in a sharp way in order to estimate the temperature gradients and the interfacial heat flux accurately. The accuracy of the phase change model is assessed with the analytical solution of the growth of a bubble in a superheated liquid. The evaporation rate of bubbles in superheated rectangular microchannels is investigated in the liquid film regime and the effects of geometry (aspect ratio) are studied. The effects of liquid superheat and operating conditions and thereby liquid film and thermal boundary layer thicknesses are discussed in terms of wall and interfacial heat fluxes.

\(^1\)TA acknowledges the Irish Research Council for the funding, ICHEC and RCS (Research Computing Service of Imperial College London) for computational resources, as well as IMFT (Dominique Legendre and Annag Pedrono) for the use of the JADIM code.

3:26PM A24.00003 Quantitative Phase Field Simulations of Turbulent Two-Phase flows\(^1\), NATHAN LAFFERTY, Paul Scherrer Institute, ARNOLDO BADILLO, OMAR MATAR, Imperial College London — Turbulent two-phase flows are ubiquitous and fundamental in our society. We find them in ordinary tasks such as cooking (boiling), but also in more complex systems such as internal combustion engines, power stations, and atmospheric flows just to name a few. A better understanding of the interaction between turbulent structures and interfaces, will contribute to the derivation of more accurate models, which can then be used in the optimization of current technologies or aid in the development of new ones. To assess the capabilities of the Phase-Field model in predicting turbulent two-phase flows, we simulate the dynamics of a Taylor bubble rising in a cylindrical pipe. The simulations lead to the formation of a complex turbulent pattern in the wake of the Taylor bubble, which is strongly coupled to the dynamics of the bubble skirt. We observed important changes in the turbulent structures, as we vary the level of resolution of the surface tension force. We compare our prediction of bubble rise velocity, mean velocity profiles and turbulent fluctuations with direct experimental measurements. We discuss our findings, in terms of a recently developed analytical expression for the physical error in the calculation of the surface tension force.

\(^1\)We acknowledge funding from the Royal Academy of Engineering (Research Chair for OKM) and EPSRC (grant EP/K003976/1).

3:39PM A24.00004 Violent Expansion of a Rising Taylor Bubble\(^1\), GUANGZHAO ZHOU, ANDREA PROSPERETTI, University of Houston — Because of the gradually decreasing hydrostatic pressure, a Taylor bubble expands as it rises in a long vertical conduit such as are encountered in volcanoes and deep-water oil drilling. In some situations, the expansion becomes violent, with a rapid increase of the bubble volume and a possibly catastrophic ejection of liquid from the mouth of the conduit. The mechanism of this process is analyzed with a 1-D drift-flux model and a simpler dynamic model. The results of the two models agree with each other. A simple but useful criterion for the occurrence of the violent expansion is obtained from a quasi-equilibrium model. The important nondimensional parameters involved in the process are identified and, on their basis, the energy budget for the rising bubble is clarified. The effect of gas diffusing from the liquid into the rising bubble is also considered and deficiencies in the current state of knowledge about this aspect of the problem are identified.

\(^1\)Research reported was supported by the Gulf Research Program of the National Academies of Sciences, Engineering, and Medicine under Award No. GRP200008864.

3:52PM A24.00005 How fast do bubbles rise in strong turbulence with a high energy dissipation rate?\(^2\), RUI NI, ASHWANTH SALIBINDLA, ASHIK ULLAH MOHAMMAD MASUK, Johns Hopkins University — We carried out an experimental study on the rise velocity of finite-size bubbles from 1 mm to 10 mm in diameter in strong turbulence. The turbulence generated in our facility targets an energy dissipation rate close to the one typically experienced in oceanic environments, which is orders-of-magnitude larger than most existing laboratory facilities. To uncover the mechanisms of bubble-turbulence interaction and its effect on the bubble rising velocity, simultaneous measurements of both the dispersed and the carrier phases in 3D were obtained. The results suggest that the bubble rising velocity shows an interesting transition: small bubbles rise much slower and large bubbles rise faster than that in the quiescent medium. In addition, the transition is at the point when bubble deformation becomes important. A simple model is introduced to quantify this transition and use it to infer both the lift and drag coefficients.

\(^2\)This research was supported by the Gulf Research Program of the National Academies of Sciences, Engineering, and Medicine under Award No. GRP200008864.
4:05PM A24.00006 Path transition of a spiraling rising bubble: a wake-controlled process by imposing magnetic fields

JIE ZHANG, School of Aerospace, Xi’an Jiaotong University, LONG CHEN, MINGJIU NI, School of Engineering Science, University of Chinese Academy of Sciences — The path transition of a spiraling bubble under the influence of magnetic fields are investigated. It is found that the rising path of a spiral bubble can be controlled manually by imposing magnetic fields in different directions and magnitudes. To detect what happens to the bubble when magnetic fields are applied, two research strategies have been adopted. First we look into details at the evolution of the wake vortices after imposing magnetic fields, and we will show the path transitions and the closely related to the wake evolutions. Second, by calculating the time histories of the forces experienced by the bubble in presence of external magnetic fields, the results also reveal how the forces and the vortex patterns are coupled during path transition. Generally, the present study aims to provide the possibility of controlling the motion of bubbles, and potentially, this possibility conveys us that the key to control the bubble motion is to reconstruct the wake vortices by changing the flow field.

4:18PM A24.00007 Laboratory Experiments on Air Bubbles Rising through Carbopol Capped with Water

KAI ZHAO, EDMUND TEDFORD, Department of Civil Engineering, University of British Columbia, MARJAN ZARE, Department of Mathematics, University of British Columbia, IAN FRIGAARD, Departments of Mathematics and Mechanical Engineering, GREGORY LAWRENCE, Department of Civil Engineering, University of British Columbia — We have conducted a series of laboratory experiments bubbling air through a Carbopol solution capped with water. These experiments were conducted to better understand methane ebullition, and its effects on turbidity, in a pit lake. The first bubble is the largest and drags Carbopol into the water. Subsequent bubbles are smaller and follow the path of the first bubble, creating a tube within the Carbopol into which water flows. For a range of Carbopol concentrations we investigate the size and shape of the bubbles, their rise trajectory, the amount of Carbopol dragged into the water, and the evolution of the tubes. Higher concentrations of Carbopol can support deeper tubes without collapsing. These tubes resemble the pothole marks observed in the pit-lake bed.

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A25: Bubbles I

3:00PM A25.00001 Shock wave interactions in single-bubble collapse near a corner

WILLIAM WHITE, SHAHABODDIN AALAHYARI BEIG, ERIC JOHNSEN, University of Michigan — Damage to neighboring surfaces due to repeated bubble collapse is one of the most important consequences of cavitation, which can be found in a multitude of hydraulic systems. A number of experimental studies have been conducted to predict the dynamics of re-entrant jets as well as bubble migration in a corner. However, the temperatures and pressures during the collapse have yet to be investigated. In this study, we quantify the effects of bubble-boundary interactions on the bubble dynamics and the temperatures/pressures produced by the collapse of a single bubble near two perpendicular rigid surfaces. For this purpose, we use an in-house, high-order accurate shock- and interface-capturing method to solve the 3D compressible Navier-Stokes equations for gas/liquid flows. The non-spherical bubble dynamics are investigated, and the subsequent pressure and temperature fields are characterized based on the relevant parameters entering the problem: geometry, stand-off distances from each surface, driving pressure. We demonstrate that bubble-boundary interactions amplify/reduce pressures and temperatures produced during the collapse and increase the collapse time and the non-linearity of the bubble displacement, depending on geometric parameters.

3:13PM A25.00002 The role of compressibility and vorticity in the collapse of a bubble near a rigid boundary

MINKI KIM, SHAHABODIN AALAHYARI BEIG, ERIC JOHNSEN, University of Michigan, Ann Arbor — Cavitation-bubble collapse is known to cause structural damage in a variety of industrial applications such as naval hydrodynamics and turbomachinery. The concentration of energy and shock emission during the non-spherical collapse is expected to depend on the liquid compressibility, and possibly the vorticity produced during the process. Thus, a better understanding of role of compressibility and vorticity is essential to predicting cavitation erosion. In this study, we compare direct simulations to potential flow calculations to extract the effects of compressibility and vorticity on the collapse of a gas bubble near a rigid boundary. The 3D compressible Navier-Stokes are solved in the gas and liquid using a high-order shock- and interface-capturing scheme; potential calculations are conducted using a boundary integral method. We observe a delay between the two approaches, attributed to the differences in the pressure fields at early times due to compressibility effects. Nevertheless, bubble morphologies are similar for most of the collapse, with discrepancies visible only in the last stage of collapse. The vorticity evolving during the collapse may play a role on the bubble dynamics at this stage.

3:26PM A25.00003 Collapse of individual bubbles near rigid boundaries

SHAHABODIN AALAHYARI BEIG, ERIC JOHNSEN, Mechanical Engineering Department, University of Michigan — Cavitation happens in a variety of applications ranging from naval hydrodynamics to biomedical ultrasound. The inertial collapse of cavitation bubbles concentrates energy into a small volume, and is capable of producing high pressures and temperatures and emitting radially propagating shock waves. One important consequence of such phenomenon is structural damage to neighboring objects following repeated collapse of cavitation bubbles. In order to provide a better understanding of this problem, we perform high-resolution numerical simulations of the inertial collapse of individual bubbles near rigid surfaces by solving the three-dimensional compressible Navier-Stokes equations for gas/liquid systems. By considering the collapse of a single bubble as well as a bubble pair near a rigid surface, we explain that the bubble-boundary and/or bubble-bubble interactions can affect the energy concentration, give rise to the kinetic energy of non-converging motions, and break the symmetry of the collapse that modify the pressures and temperatures thereby produced. The results are used to investigate the non-spherical bubble dynamics and characterize the pressure and temperature fields based on the relevant parameters entering the problem: initial stand-off distance, distance and angle between the two bubbles, and driving pressure This work was supported by ONR grant N00014-12-1-0751 under Dr. Ki-Han Kim.

3:39PM A25.00004 ABSTRACT WITHDRAWN –
3:52PM A25.00005 The diameters and velocities of the jet droplets produced after bubble bursting, FRANCISCO J. BLANCO-RODRIGUEZ, JOSE M. GORDILLO, Universidad de Sevilla — Here we provide a theoretical framework revealing that the radius \( R_0 \) of the top droplet ejected from a bursting bubble of radius \( R_0 \) can be expressed as

\[
R_0 = \frac{0.22 R_0}{1 - \left( \frac{2\mu}{\rho R_0^2} \right)^{1/2}}
\]

for \( Oh \ll 0.3 \) and \( Bo \ll 1 \) with \( Oh = \mu/\sqrt{\rho\sigma} \ll 1 \) the Ohnesorge number, \( Bo = \rho g R_0^2/\sigma \) the Bond number and \( \rho, \mu, \) and \( \sigma \) the liquid density, viscosity, and surface tension coefficient respectively. This prediction, which agrees very well with both experimental data and numerical simulations for all the values of \( Oh \) and \( Bo \) investigated, can be particularized to express the diameters of the jet droplets produced from the bursting of sea bubbles with radii \( R_0 \leq 1 \) mm, with implications in marine aerosol production. The velocities of the first drops ejected are also expressed as a function of \( Oh \) and \( Bo \), being this initial drop velocity largely reduced by air drag at tiny distances \( \sim R_0 \) above the interface. We find that the ratio between the radius of curvature at the tip of the jet and the jet radius controls the growth of capillary instabilities, a fact explaining why no droplets are ejected from the tip of the fast Worthington jet for values of \( Oh \) slightly larger than \( Oh^{0.3} \).

---

Saturday, November 23, 2019 3:00PM - 4:31PM –
Session A26 Experimental Measurements: General - 608 -

3:00PM A26.00001 Micro-PIV Evaluation of Lateral Flow Assays, EMILIE NEWSHAM, STEVE WERELEY, JACQUELINE LINNES, Purdue University — Lateral flow assays (LFAs) are an increasingly common technology (e.g. pregnancy tests) but are limited in scope and sensitivity due to minimal understanding of their underlying fluidic properties. LFAs comprise porous paper membranes that transport liquid samples along a test strip through capillary action to dried reagents, where binding reactions produce a readable signal. Mathematical models informed by analytical evaluations have been created to optimize LFA development, but none use empirical microfluidic data to derive model parameters. Here, micro-PIV is used to evaluate liquid flow in LFAs. Videos of liquids carrying nanoparticles flowing through LFA membranes under different conditions are analyzed using micro-PIV. The porous membrane has different permeability to the liquid and nanoparticles which means the observed particle motion is not the same as the fluid motion. A scale factor between the visible macroscopic fluid front and microscopic particle velocity is derived. This novel microscopic evaluation method will inform intelligent models for more efficient development of LFAs.

3:13PM A26.00002 V3V Measurements on a Propeller in a Cavitation Water Channel, DAN TROOLIN, WING LAI, TSI Incorporated — Cavitation around ship propellers occurs due to low pressure regions where cavitation bubbles form and collapse potentially causing severe damage to propeller blade surfaces through unequal loading, vibration, and pitting. In order to avoid cavitation damage care must be taken in the design, selection and operation of the propeller blades. A study was conducted in the West Japan Fluid Engineering Laboratory Co., Ltd. closed return cavitation water channel with a test section of 500 500 2000 mm of the flow downstream of a five-bladed, powered aquatic propeller at a freestream velocity of 4.0 m/s. A V3VFlex volumetric measurement system with four 4 megapixel cameras was used in order to provide high-resolution instantaneous and phase-averaged volumetric measurements. The measurements reveal the presence of vortex structures, locations and interactions. Results of the propeller wake will be presented and examined.

3:26PM A26.00003 Gridded Analysis of Volumetric Particle Velocity Data, MATT STEGMEIR, TSI Incorporated — 2-Frame and time-resolved Volumetric Particle Tracking has become an increasingly valuable tool for 3-D flow characterization. Unstructured, randomly-distributed data are typically produced by tracking individual particle locations either between 2 sequential image frames or by constructing longer Lagrangian particle position tracks to from highly time-resolved data. High-resolution datasets are typically interpolated onto an ordered grid in order to provide greater convenience in data presentation and to facilitate calculation of derived quantities. Interpolation schemes are compared for accuracy and resolution, and robustness to spurious data points. Improvements are proposed to provide for increased resilience against noise and spurious data points in experimental data sets. Results are compared to standard Gaussian-weighted interpolation. Single- realization and statistical results are presented from analysis of synthetic and experimental datasets.
3:39PM A26.00004 Droplet sizing of opaque liquid with integrated transmitting and receiving optical arrangement. WING T. LAI, DANI TROOLIN, TSI Incorporated — A number of industrial applications require the precise sizing of droplets. Some examples are spray drying in drug formation and ink jet printing for TV panels. The technique of ink jet printing for making TV panels requires the droplets emitting from the printer nozzles to be monodisperse and uniform for a long period of time. A typical printer head consists of multiple nozzles, from 256 to 1046. Every nozzle is designed to be producing the same droplet size of liquid over the period of time when the panel is being made. Hence it is important to be able to monitor all the nozzles regularly to ensure that the uniform droplets are produced. If there is any size deviation of the droplets from the nozzles, the quality of the panel can be substantially degraded, making it non-useable. An integrated transmitting and receiving optical probe was designed specifically for the measurement of the droplets from all the nozzles in a sequential fashion such that every nozzle in the printer head can be characterized. The droplet result provides the feedback of any faulty nozzle behavior (one not giving the correct droplet size) and adjustment of the flow system can be made to get the nozzle working normal again.

3:52PM A26.00005 Development of a Flexible Four-Camera Volumetric PIV System for Tow Tank Applications. RUBEN HORTENSIIUS, STAMATIOS POTHOS, MARK CECCONI, TSI Incorporated — Traditionally PIV experimentation for hydrodynamic, marine, and biological applications, have been carried out in water tunnel facilities, where the instrumentation is safely located outside of the facility. Due to practical constraints, this has often resulted in limited ability to conduct experiments at large scale or at high magnifications. To overcome this, submersible PIV systems have been developed, allowing for PIV experimentation in facilities such as towing tanks. These tow tank PIV systems, whether 2D/planar-PIV or 3D/stereo-PIV, are still regarded as very unique and highly specialized, in no small part due to the exceptionally distinctive facilities which house them. TSI has provided numerous PIV systems for underwater applications in the past, and in 2017 produced a specialized system capable of being used for 2D/planar measurements, 3D/stereoscopic measurements, and 3D3C volumetric measurements. Details of the capabilities of the new system are described, as well as insights gleaned from the design, manufacture, and installation of this one-of-a-kind system. Preliminary volumetric PIV results obtained with the system are shared and discussed.

4:05PM A26.00006 Volumetric 3D-3C (V3V) Particle Tracking Velocimetry measurement of turbulent twin jet. NIMESH VIRANI, VESELINA ROUSSINOVA, Department of Mechanical, Automotive and Materials Engineering, University of Windsor, RAM BALACHANDAR, Department of Civil and Environmental Engineering, University of Windsor — Recently developed (by TSI, Inc., Shoreview, MN, USA) V3V: Volumetric three-direction, three-component (3D-3C) particle tracking velocimetry technique, is applied for understanding of inherently complex interaction between turbulent twin jets. Two jets are separated by a distance of approximately 2D, where D is the diameter of the jet and Reynolds number based on exit diameter of the pipe, for both jetting velocimetry technique, is applied for understanding of inherently complex interaction between turbulent twin jets. Two jets are separated by a distance of approximately 2D, where D is the diameter of the jet and Reynolds number based on exit diameter of the pipe, for both jets is around 12000. Jets are released into a channel comprising of quiescent water and channel dimensions are such that, jets can remain free from any kind of boundary effects. The measurement volume of approximately 15010050 mm^3 in X, Y and Z directions, which is equivalent to 63D42D21D in terms of pipe diameter, was investigated for mean and turbulence characteristics of problem under consideration. Particle tracking velocimetry (PTV) is applied to trace randomly spaced seeding particles between concurrently captured two frames of the same instant of flow field. PTV eliminates inherent bias present in the traditional FFT correlation based analysis of particle image velocimetry (PIV). Access to all three components of velocity enabled us to get insight of shear layer merging phenomena between two jets, which resulted into some interesting instantaneous flow structures. Time-averaged and turbulent characteristics of jets’ interaction are analyzed using various statistical analysis tools and they reiterate the necessity of 3D examination.

4:18PM A26.00007 Experimental Characterization of Flow Induced by a Nanosecond Surface Discharge. LALIT RAJENDRAN, BHAVINI SINGH, RAVICHERA JAGANNATH, GEORGE SCHMIDT, PAVLOS VLACHOS, SALLY BANE, Purdue University — Nanosecond surface Dielectric Barrier Discharges (ns-DBD) have generated growing interest as a means of high speed flow control. These discharges are characterized by electrical breakdown caused by high voltage, nanosecond pulses resulting in ultra-fast heating of the surrounding air. The rapid heat release leads to generation of a shock wave and complex flow characterized by coherent vorticity and a hot gas kernel near the electrode surface. Past applications of these discharges have yielded mixed results in flow control and the reasons for success/failure of these actuators are not well understood. This is because a fundamental understanding of the induced flow field and its relation to electrode geometry and energy deposited is not available. To address this limitation, we have performed high speed Particle Image Velocimetry (PIV) and Background Oriented Schlieren (BOS) measurements on the flow induced by a single nanosecond filamentary surface discharge under quiescent conditions. We measure the vorticity and density of the hot gas kernel and create a mechanistic model of the actuator induced flow. Such a model of the actuator performance can inform the choice and deployment of these devices for flow control applications.

1This material is based upon work supported by the U.S. Department of Energy, Office of Science, Office of Fusion Energy Sciences under Award Number DE-SC0018156

Saturday, November 23, 2019 3:00PM - 4:31PM – Session A27 Biological Fluid Dynamics: Flight 609 - Geoffrey Spedding, USC

3:00PM A27.00001 Unsteady aerodynamic characteristics of large scale bird flapping flight. ZIFENG WENG, School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai, 200240, China, SUYANG QIN, YANG XIANG, HONG LIU, J. C. Wu Center for Aerodynamics, School of Aeronautics and Astronautics, Shanghai Jiao Tong University, Shanghai, 200240, China — Various unsteady aerodynamic mechanisms have been found in flapping flight especially at low Reynolds number (1E2-1E4). However, large scale bird flying at high Re (1E5) is mostly assumed to be quasi-steady. A two-jointed-wing robotic goose was built to carry out wind tunnel experiment. Force and PIV measurements were performed to study the unsteady force and flow field. Lift is found to change with average effective angle of attack and the average lift is close to that in cruising flight. Lift enhancement occurs in mid-downstroke, which is not induced by leading edge vortex but the result of enhancement of circulation. Thrust occurs in both upstroke and downstroke periods, and maximizes at mid-upstroke and mid-downstroke. Since flapping amplitude varies along wing span, thrust production is the overall result at different spanwise position. According to the flow field measurement, jet flow is found in the wake of handwing, but not in that of armingwing. The phase difference between handwing and armingwing results in larger flapping amplitude, i.e. higher St and thus enhances thrust production. In general, high Re flapping flight play its role mostly in propulsion, instead of producing high lift as that in low Re flight. And lift production might rely more on aerofoil and flow speed.
3:13PM A27.00002 Simulation of flapping bird flight, part 1: closed-loop control, forces, and wake topology1, VICTOR COLOGNESI, GIANMARCO DUCCI, RENAUD RONSSSE, PHILIPPE CHATELAIN, UCLouvain, REVEALFLIGHT TEAM — This work aims at reproducing bird flight in silico in order to shed light on its performance-enabling mechanisms. To that end, we establish an anatomical model of a bird with a pure flapping gait, the bald ibis. It combines a multi-body model of the skeleton, with the corresponding joints and degrees of freedom, and a plumage model. The latter allows extracting the wing geometry and aerodynamic properties for any configuration of the skeleton. These properties are translated into a deforming immersed lifting line, which handles the sources of vorticity within a vortex particle-mesh method. The resulting multiphysics framework captures, at a high fidelity, the full flight dynamics, the required efforts and the resulting wake. A sensitivity analysis is carried out and leveraged for the design of a flight controller. Specifically, we quantify the influence of shoulder kinematic features on the aerodynamic forces and moments. The controller then uses these features as command parameters. This bird model can reach a trimmed state over various flight regimes and can handle transients between them. Finally, the accurate capture of the wake vortical structures allows their unambiguous identification and association with the time-varying aerodynamic forces produced by the bird.

1This project was supported by the ARC program of the Federation Wallonie-Bruxelles (grant number 17/22-080, REVEALFLIGHT)

3:26PM A27.00003 Simulation of flapping bird flight, part 2: Gait parametrization, limit cycle, and dynamic stability1, GIANMARCO DUCCI, VICTOR COLOGNESI, PHILIPPE CHATELAIN, RENAUD RONSSSE, Universite catholique de Louvain, REVEALFLIGHT TEAM — We introduce a method to identify gait trajectories in trimmed flapping flight. Such trajectories correspond to limit cycles in the state space, characterized by the same period as the wingbeat. Our method relies on a multiple-shooting algorithm that can simultaneously identify unknown limit cycles, and analyze their stability. Based on Floquet theory, this analysis computes the Jacobian of the identified limit cycle and assesses its stability from its eigenvalues.

In a first contribution, we adapt this framework to the flapping flight equations of motion, known to be not only non-linear and time-dependent, but also driven by state-dependent forcing aerodynamic loads. An aerodynamic model was developed following the quasi-steady lifting line approach reported in part 1, taking the wing morphology and prescribed kinematics as input, and returning the state-dependent aerodynamic loads as output.

Our results identify one instable mode, suggesting that birds continuously rely on sensory feedback to achieve steady-state flapping flight. This framework is then leveraged in the analysis of several gait configurations. In particular, we use it to perform a sensitivity analysis of the flapping gaits required to achieve several flight regimes (level, climbing and descending flight).

1This project was supported by the ARC program of the Federation Wallonie-Bruxelles (grant number 17/22-080, REVEALFLIGHT)

3:39PM A27.00004 ABSTRACT WITHDRAWN —

3:52PM A27.00005 Holey wings can improve aerodynamics at bioscales, YOHANNA HANNA, GEOFFREY SPEDDING, University of Southern California — The performance of wings at moderate Reynolds numbers (10^3 \leq Re \leq 10^5) is strongly influenced by laminar boundary layer separation, and by the possible reattachment (in the mean sense) of the detached shear layer. Details of these events on both pressure and suction surfaces can lead to unexpected phenomena such as a negative lift slope around zero angle of attack, and abrupt changes in flow state close to the formation of a closed laminar separation bubble. Once understood, one may seek to exploit these sensitivities to find new forms of flow control, either passive or active. Here, we show that small, distributed porosity (porosity ratio, \phi = 0.005) on a wing at moderate Re can almost completely remove the nonlinearities in the C_l(\alpha) curve, to yield a more robust and predictable lifting device. The flow-through mechanisms of a permeable wing are investigated and provide an alternative explanation for the sometimes-reported benefits in bio-flyers, that would then also apply to engineered equivalents of the same scale.

4:05PM A27.00006 Arrow flight and optimal feathers in archery, TOM MADDALENA, CAROLINE COHEN, CHRISTOPHE CLANET, LaHyX, Ecole polytechnique — In archery, athletes shoot arrows with their bow from 70 m on an outdoor field. The target is made of 10 concentric rings, from 12 cm to 1.20 m. Therefore, the angular tolerance of the smallest ring is 0.12/70 = 2 mrad. What is more, the environmental conditions such as wind or rain can have a huge impact on the arrows final position. Yet, a few centimeters on the target often distinguishes the winner from the others. All parameters of the bow and arrows must then be carefully selected in order to maximize the accuracy. In this work, we focus on the arrows flight, and more particularly on the choice of the optimal feathers. Archers can indeed choose among many size and shapes of feathers, and the effect of those feathers on the accuracy is still poorly understood. To tackle this question, we combine different experiments. We first shoot arrows with feathers of different size and shape with a throwing machine. Those experiments are eventually achieved with a lateral wind. We also characterize in a wind tunnel the influence of the feathers on the aerodynamic forces on the arrow. Those experiments combined with a theoretical model of the flight, allow to predict the trajectory of an arrow, and the influence of different perturbation on its final position.

4:18PM A27.00007 Falling of a plate with eccentric center of mass, SEAN COUGHENOUR, HUI WAN, University of Colorado, Colorado Springs — The problem of a freely falling object is of interest in both fluid mechanics and nonlinear dynamics. Four types of motions have been observed, i.e., steady falling, flutter, tumble, and chaotic motion, depending on the body moment of inertia and the Reynolds number. For example, at high Reynolds number, flutter and tumble motion are obtained for bodies with small and large moment of inertia, respectively. Most of previous studies focused on the symmetric body with mass center collocated with the geometric center. In this study, we will numerically study the falling of a plate with eccentric center of mass. We will also adjust the center of the mass and thus change the moment of inertia and the angular velocity of body rotation, therefore enabling a new methodology to actively control the plate falling trajectories.
3:00PM A28.00001 Interaction Between Fibers and Viscous Flow Evaluated by Simplified Bead-Chain Model and Generalized SPH. YOSHIKAKI ABE, TOMONAGA OKABE, Tohoku University — This study investigates the motion of short fibers in melted resin for a three-dimensional (3D) printing process. We developed a numerical model for fiber and viscous-flow interaction problems based on the generalized smoothed particle hydrodynamics (generalized SPH) and the simplified bead-chain model (SBCM) for short fibers. The generalized SPH allocates particles inhomogeneously in a physical space and arranges them via mapping in a generalized coordinate system where the particles are aligned at a uniform spacing, which allows efficient simulation compared with conventional Cartesian SPH approaches. The SBCM models each fiber as a chain of spheres given by a single equation of translational motion. The effects of bending and hydrodynamic torques, which significantly simplify the governing equations and results in a more cost-effective simulation than conventional models such as the BCM or classical microstructure-based fiber suspension model. This study proposes the SBCM-SPH model for predicting a fiber-flow interaction and investigates the motion of short fibers in the fused decomposition modeling process of 3D printer.

3:13PM A28.00002 The Quest For Enhanced Pulsed Slurry Atomization Via Modulation. WAYNE STRASSER, Liberty University — Our non-Newtonian airblast atomization flowfield violently pulses (axially and radially) by self-generating and self-sustaining interfacial instability mechanisms. Recent work demonstrated that exothermic chemical reactions enhance this moderate Mach number atomization. Explored herein is the potential to further enhance reaction-assisted disintegration by superimposing forced mass flow variations onto otherwise constant gas feed streams. Two nozzle geometries (high and low prefilming distance) and multiple superimposed feed frequencies are considered for each gas stream. Results indicate that superimposed frequencies have potential to enhance chaotic atomization in a statistically significant manner. Ironically, the most efficient atomization did not coincide with the highest levels of gas resonance. A detailed study of reactor start-up flow reveals new mechanisms which explain performance differences. The prefilming design facilitates an isolated mixing region where secondary ignition occurs nearly immediately after startup and augments reaction product conversion.

3:26PM A28.00003 A novel selective lithium separation using amorphous pMOF/alginate interfaces. SUNG HO PARK, SANG JOON LEE, Pohang University of Science and Technology — Anomalous opposite ion behaviors of Li$^+$ and Mg$^{2+}$ ions are experimentally observed from amorphous interfaces of phosphonate metal–organic framework (pMOF)/alginate composites. The amorphous structures of pMOF/alginate composites are significantly varied according to reaction temperature and intertwinement degree of alginate networks. As a proof-of-concept application, Li$^+$ and Mg$^{2+}$ ions are separated using pMOF/alginate composite depending on the degree of pMOF growth. Initially, lots of Li$^+$ ions are effectively attracted compared to Mg$^{2+}$ ions due to the strong repulsion force of Al$^{3+}$ ions in amorphous alginate interfaces on multi-valent metal ions. However, amorphous pMOF/alginate interfaces induce effective rejection of Li$^+$ ions with low hydration energy through dehydration due to the significant interaction with water molecules, while Mg$^{2+}$ ions with high hydration energy are significantly attracted by negatively charged phosphonate groups. The present results of amorphous MOF/alginate interfaces would provide a variety of benefiting separation opportunities with unique property in seawater desalination and rare metal recovery.

1This research was supported by National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (NRF-2017R1A2B3005415).

3:39PM A28.00004 The impact of monovalent and divalent ions on the viscosity of a solution with silica nanoparticles. SAHEED OLAWALE OLAYIWOLO, MORTEZA DEJAM, Department of Petroleum Engineering, University of Wyoming, 1000 E. University Avenue, Laramie, WY 82071-2000, USA — Nanoparticles (NPs) are injected into the reservoirs for enhanced oil recovery. Several mechanisms like wettability alteration, reduction in the interfacial tension, and change in the rheological properties of the fluid are involved in this process. The rheological properties of the fluid depend on the interaction between the ionic components of the brine and the injected silica NPs. As a result, the flow is affected by the interaction of NPs and salt ions. The viscosity of fluid reduces with an increase in the temperature but the impact of the ionic components of brine on the viscosity of nanofluid is rarely investigated. Therefore, this study focuses on the effect of ionic components of brine (Na$^+$, K$^+$, Ca$^{2+}$, Mg$^{2+}$, Cl$^-$, and SO$_4^{2-}$) at different concentrations of silica NPs (0.01-1 wt%) on the viscosity of fluid. It was observed from the experimental data that the viscosity of fluid is increased by the divalent anions while it is reduced by the divalent cations. The interaction between the ions of NPs and the salt ions, which causes the variation in the viscosity of fluid, is further investigated by measuring the zeta potential and the particle size distributions.

1The financial support from the Department of Petroleum Engineering in the College of Engineering and Applied Science at the University of Wyoming is gratefully appreciated.

3:52PM A28.00005 Unsteady fragmentation upon drop impact: Sheet dynamics. Y. WANG, L. BOUROUIBA, MIT — Prior work, Wang & Bourouiba 2018b, showed that upon drop impact on a finite surface, continuous secondary droplet shedding occurs, and that both size and speed distributions of the droplets ejected are governed by the unsteadiness of the sheet expansion. In turn, this continuous shedding influences the sheet expansion. Incorporating continuous droplet shedding, we show how the sheet is governed by a non-Galilean expansion law from which we predict the time evolution of all key physical quantities from the sheet radius to the fluid shed by the rim, all in agreement with experimental measurements. We also discuss a peculiar property of the governing equation which imposes a time-to-maximum-radius independent of the impact energy. This property results in a temporal evolution of the partition of mass, momentum, and energy in the sub-parts of the fragmenting system that is independent of the impact conditions. We discuss the robustness of the results to changes of fluid properties.

4:05PM A28.00006 Unsteady fragmentation upon drop impact: prediction of droplet size and speed distributions. L. BOUROUIBA, Y. WANG, MIT — Upon impact on a finite surface, a drop first expands into a sheet in the air, surrounded by a rim that destabilizes into ligaments that, in turn, shed secondary droplets. Wang & Bourouiba 2018b and Wang et al. 2018 showed that both size and speed distributions of the secondary droplets ejected during fragmentation are shaped by the unsteadiness of the sheet and rim. In this combined experimental and theoretical study, we derive and validate the analytic expression governing both size and speed distributions of secondary droplets ejected during such unsteady fragmentation, including time-evolution of the mean quantities associated with the distributions. We discuss the implications for various applications including contamination dispersal.
algorithm to classify between normal and moderate OSAS patients. This algorithm classification showed about 80% classification accuracy that
unknown patients. This method can eliminate the high time consuming of CFD computation time. Also, we use Support vector machine learning
increasing the quantity of airway model data and simulate it to get aerodynamic features. Because of the requiring of heavy computational time
the computed tomography (CT) data. The patients who have sleep apnea have considerable pressure drop occurred due to a narrow airway.
method to patent. In order to solve these problems, people try to use computational fluid dynamics (CFD) using upper airway geometry from
(PSG), however, have not been able to provide quantitative criteria and the non-scientific judgment method. Moreover, it is uncomfortable
the number of patients who undergo orthodontic treatment of sleep apnea is increasing. Existing diagnostic methods, the polysomnography
YOUNG WOO KIM, JOON SANG LEE, HYUNG JU CHO, YOON JEONG CHOI, HWI DONG JUNG, Yonsei university, Seoul, Korea — Recently,
air can accumulate in the respiratory organs in the human body and cause various types of diseases. Various types of masks for filtering fine dust are
commercially available. Given the trade-off between the filtering performance and wearing comfort, the physical understanding of respiratory
mechanics through a mask is important in assessing the performance of the masks in mask design. We suggest an experimental setup to
evaluate the filtering performance and wearing comfort of dust masks. By constructing a respiration simulator, we measure dust particles that are
filtered by a mask and quantify the respiratory resistance in terms of the power required for respiration. We analyze the effects of additional
elements such as fans and valves attached to the mask to assist respiration.

3:39PM A29.00004 Computational analysis of airflow dynamics for predicting collapsible sites in the upper airways: Advanced machine learning
SUSIE RYU, SEUNG HO YEOM, YOUNG WOO KIM, JOON SANG LEE, HYUNG JU CHOI, YOON JEONG CHOI, HWI DONG JUNG, Yonsei university, Seoul, Korea — Recently, the number of patients who undergo orthodontic treatment of sleep apnea is increasing. Existing diagnostic methods, the polysomnography (PSG), however, have not been able to provide quantitative criteria and the non-scientific judgment method. Moreover, it is uncomfortable method to patent. In order to solve these problems, people try to use computational fluid dynamics (CFD) using upper airway geometry from the computed tomography (CT) data. The patients who have sleep apnea have considerable pressure drop occurred due to a narrow airway. This drop can be measured by using a pressure sensor placed on the airway. CFD modeling is used to predict the airflow through the airway for increasing the quantity of airway model data and simulate it to get aerodynamic features. Because of the requiring of heavy computational time cost, this study uses a machine learning algorithm. We use multivariate gaussian process machine learning for predicting aerodynamic features of unknown patients. This method can eliminate the high time consuming of CFD computation time. Also, we use Support vector machine learning algorithm to classify between normal and moderate OSAS patient. This algorithm classification showed about 80% classification accuracy that can be useful decision to clinician.

3:52PM A29.00005 Elasto-capillary network dynamics of inhalation
FELIX KRATZ, JEAN-FRANCOIS LOUF, Department of Chemical and Biological Engineering, Princeton University, ANVITHA SUDHAKAR, None, NATHANIEL JI, SUJIT DATT, Department of Chemical and Biological Engineering, Princeton University — The seemingly simple process of inhalation relies on the complex interplay between muscular contraction in the thorax, elasto-capillary interactions in the individual airway branches, connectivity between different branches, and overall air flow into the lungs. Sophisticated pulmonary fluid dynamics models have been developed to elaborate the competition between capillarity, which tends to keep flexible branches closed, and elasticity, which favors opening, for single airway branches. However, a quantitative model combining the physiological opening process of flexible airway branches with the biomechanics and interconnected geometry of the lungs is still missing. To address this issue, we develop a statistical model of the lungs as a symmetrically-branched network of liquid-filled cavities, interfaced to a viscoelastic network that represents the airway. The opening and closing of the airway is determined by input biomechanical parameters, enabling us to test the influence of changes in the mechanical properties of lung tissues and secretions on inhalation dynamics. By summing the dynamics of all the individual branches, we quantify the evolution of overall lung pressure and volume during inhalation, and find good agreement with typical breathing curves obtained in the literature.
3:00PM A30.00001 3D Confinement Effects on Helicobacter pylori Swimming, SURAJ KUMAR KAMARAPU, HENRY FU, University of Utah — H. pylori bacterium has evolved to swim through highly acidic gastric mucus layer by diffusing ammonia from its body, neutralizing the surrounding medium and forming a pocket of Newtonian fluid around itself. The shape of this pocket, which depends on the Peclet number, determines the overall swimming behavior of the bacterium. We previously used a 2D Taylor sheet model to study the swimming bacterium near a Brinkman medium that represented the mucus gel and found that the swimming speed monotonically increases as the distance between the swimmer and the gel decreases. However, swimming in such situations can also be highly dependent on boundary geometry, diffusion, swimming flows around the swimmer and requires a complete 3D model factoring in the above possible influences. Here we model the mucus gel with a random spatial distribution of regularized stokelets placed outside the Newtonian fluid pocket, and quantify its influence on the swimming speeds for a constant stroke. Advection-diffusion of ammonia is treated numerically allowing us to access large Peclet numbers. We find that for small Peclet numbers, the bacterium swims faster than predicted by a 2D model, but beyond a certain Peclet number, the bacteria can swim with reduced speed as it faces 3D confinement upstream.

3:13PM A30.00002 Motility of flagellated bacteria in colloidal media1, SHASHANK KAMDAR2, LORRAINE F. FRANCIS, XIANG CHENG, Department of Chemical and Materials Science, University of Minnesota Twin Cities — Recent years have seen increasing interests in understanding the mechanism and motility of microswimmers in non-Newtonian fluids due to their relevance in biological and biomedical applications. Nevertheless, despite extensive study on the locomotion of microswimmers in polymeric fluids, their motion in a colloidal suspension remains largely unexplored. Here, we study the motility of E. coli, a flagellated bacterium in colloidal medium. We systematically vary the size of colloidal particles from 50 nm to 1 μm and the volume fraction up to 20%. The motion of fluorescent-labeled bacteria is imaged using confocal microscopy and speeds of bacteria are extracted using a robust in-house tracking algorithm. Our results show that bacterial motility decreases with increasing volume fractions but remains constant beyond a critical volume fraction. In addition, we find that the motility depends on the size of passive colloid. Finally, we construct a simple model that qualitatively explains our experimental observation. This work enriches the current understanding of microswimmers’ locomotion in complex fluids.

3:26PM A30.00003 How Azimuthal Swirl Impacts Swimming Kinematics in a Viscoelastic Fluid1, JEREMY P. BINAGIA, Stanford University, ARDELLA PHOA, Santa Clara University, ERIC S. G. SHAQFEH, Stanford University — Microorganisms are often found moving through viscoelastic environments such as mucus layers or biofilms. In 2014, Zhu & Lauga simulated the steady motion of a spherical “squirmmer” in a viscoelastic fluid to understand how fluid elasticity impacts the organism’s speed. The squirmmer considers a spherical swimmer that includes a specified slip velocity at its surface. This model has been used extensively to study the motion of ciliates like Paramecium, colonies of the green algae Volvox, or as a simplified model for general swimmers like E. coli. In all cases, they found that a squirmmer swims slower than it does in a Newtonian fluid. In that study and many others that use the squirmmer model, only the first two axisymmetric swimming modes are considered. Only very recently have authors considered the addition of other modes, such as those that involve azimuthal surface velocities. Recently, we have conducted simulations showing that particular combinations of the axisymmetric swirling modes can actually lead to a speed increase in an elastic fluid. In this talk, we will describe how the inclusion of this azimuthal swirl affects swimming kinematics in elastic fluids, with a focus on how polymer deformation leads to changes in speed.

1This research is supported by the UMN IPRIME program.
2Membership Pending

3:40PM A29.00007 Analysis of Inlet Velocity Profile Effects on Airflow Simulations in Patient-Specific Healthy Trachea, BIPIN TIWARI, Department of Aerospace Engineering, Auburn University, TARUN KORE, Department of Chemical Engineering, Auburn University, ZHENGLUN ALAN WEI, Wallace H. Coulter School of Biomedical Engineering, Georgia Institute of Technology, SANDEEP BODDULURI, SURYA P. BHATT, Department of Medicine, University of Alabama at Birmingham, VRISHANK RAGHAV, Department of Aerospace Engineering, Auburn University — Expiratory Central Airway Collapse (ECAC), defined by greater than 50% collapse of the trachea during expiration, is a disorder associated with Chronic Obstructive Pulmonary Disease. Pathophysiology of ECAC is multifactorial and the biofluid mechanics of airflow in the trachea could be an important factor resulting in the progression of the disease. Using computational methodology, a comprehensive investigation of the biofluid mechanics in the healthy and diseased patient-specific trachea can be conducted. One of the key considerations for setting up computations is choosing correct boundary conditions (BC). Most common BCs used by previous studies are a) flat, b) parabolic, c) Womersley, d) parabolic with an extension, and e) real, patient-specific profile. This is the first step in that direction to explore the effects of different inlet BCs for patient-specific trachea flow simulations. We test for steady and tidal flow combined with the five aforementioned inlet velocity profile conditions. Metrics such as wall shear stress and time-averaged wall shear stress were used to quantify the differences among different inlet velocity profile condition. This will lay a solid foundation towards obtaining accurate computational results in modeling ECAC.

Saturday, November 23, 2019 3:00PM - 4:31PM —
Session A30 Biological Fluid Dynamics : Micro-Swimmer General I

3:05PM A29.00006 Effect of Cartilaginous Rings in a Model of Human Trachea with Stenosis, HUMBERTO BOCANEGRA EVANS, JOS MONTOYA SEGNNINI, ALI DOOSTTALAB, Purdue University, JOEHASSIN CORDERO, Texas Tech University Health Sciences Center, LUCIANO CASTILLO, Purdue University — The human trachea is structurally supported by a series of cartilaginous rings, which introduce corrugations along the wall surface. Nevertheless, the airway walls are generally considered smooth for respiratory fluid dynamics research purposes. Previous results by Bocanegra Evans and Castillo (J. Biomech., 49, 1601, 2016) demonstrate that these rings have a significant impact on the flow separation found in the tracheobronchial bifurcation. Here, we study the effect that such rings have in a trachea model with stenosis. We present an experimental comparison of smooth and ‘ringed’ models with a grade III (70% blockage) stenotic contraction. Particle image velocimetry measurements are carried out in a refractive index-matching facility simulating resting breathing state conditions (ReD = 3,350). Our results show that cartilaginous rings induce velocity fluctuations in the downstream flow, which enhances the near-wall momentum flux and reduces flow separation after the stenosis. The maximum upstream velocity of the recirculation is reduced by 38% in the model with rings, resulting in a weaker recirculation zone. These results highlight the importance of the cartilaginous rings—and other small features—in respiratory flows.

3:13PM A30.00003 How Azimuthal Swirl Impacts Swimming Kinematics in a Viscoelastic Fluid1, JEREMY P. BINAGIA, Stanford University, ARDELLA PHOA, Santa Clara University, ERIC S. G. SHAQFEH, Stanford University — Microorganisms are often found moving through viscoelastic environments such as mucus layers or biofilms. In 2014, Zhu & Lauga simulated the steady motion of a spherical “squirmmer” in a viscoelastic fluid to understand how fluid elasticity impacts the organism’s speed. The squirmmer considers a spherical swimmer that includes a specified slip velocity at its surface. This model has been used extensively to study the motion of ciliates like Paramecium, colonies of the green algae Volvox, or as a simplified model for general swimmers like E. coli. In all cases, they found that a squirmmer swims slower than it does in a Newtonian fluid. In that study and many others that use the squirmmer model, only the first two axisymmetric swimming modes are considered. Only very recently have authors considered the addition of other modes, such as those that involve azimuthal surface velocities. Recently, we have conducted simulations showing that particular combinations of the axisymmetric swirling modes can actually lead to a speed increase in an elastic fluid. In this talk, we will describe how the inclusion of this azimuthal swirl affects swimming kinematics in elastic fluids, with a focus on how polymer deformation leads to changes in speed.

1This research is supported by the National Science Foundation (NSF) through the National Science Foundation Graduate Research Fellowship Program (NSF GRFP).
3:39PM A30.00004 Effective diffusivity of microswimmers in a crowded environment

- MARVIN BRUN-COSME-BRUNY, Grenoble Alpes University — The effect of crowded environments on micro-swimmers is studied using the micro-alga Chlamydomonas Reinhardtii (CR) as a model system. Performing a Run-and-Tumble motion in bulk, its swimming describes a persistent random walk characterized by an effective diffusion coefficient for the large-time dynamics. This swimming is experimentally observed in a complex medium made of series of pillars designed in a regular lattice, using soft lithography microfabrication. Their trajectories are tracked and analyzed. The measure of relevant statistical observables provides insight into the bias induced by the obstacles. Particularly, the mean correlation time of direction and the effective diffusion coefficient are shown to decrease when increasing the density of pillars. This provides some bases of understanding for active matter in complex environments.

3:52PM A30.00005 Modeling helical swimming in shear-thinning fluids

- NOAH LORDI, Santa Clara University, EBRU DEMIR, Beijing Computational Science Research Center, Santa Clara University, YANG DING, Beijing Computational Science Research Center, ON SHUN PAK, Santa Clara University — Swimming bacteria, such as Escherichia coli, propel by rotating their helical flagella with rotary motors in the cell membrane. Helical microswimmers rotated by external magnetic fields have been fabricated to mimic the helical propulsion of bacteria. While helical propulsion has already been extensively studied with the Newtonian fluid assumption, the performance of this propulsion mechanism in non-Newtonian fluids has attracted considerable attention recently. Biological and synthetic microswimmers move through complex fluids that often display shear-thinning viscosity. In this talk, we will discuss a theoretical model to investigate the effect of shear-thinning rheology on helical propulsion. We will highlight the similarities and differences in the propulsion performance in contrast to the results in the Newtonian limit, and compare model predictions with recent experiments on helical propulsion in shear-thinning fluids.

4:05PM A30.00006 Active particles in viscosity gradients

- GWYN ELFRING, CHARU DATT, University of British Columbia — Microswimmers in nature often experience spatial gradients of viscosity. In this work we develop theoretical results for the dynamics of active particles, biological or otherwise, swimming through viscosity gradients. We model the active particles (or microswimmers) using the squirmer model, and show how the effects of viscosity gradients depend on the swimming gait of the swimmers and how viscosity gradients lead to viscotaxis for squirmers. We also show how such gradients in viscosity may be used to sort and control swimmers based on their swimming style.

4:18PM A30.00007 The Dispersal of Swimming Microalgae in Viscosity Gradients

- MICHAEL R. STEHNACH, NICOLAS WAISBORD, JEFFREY S. GUASTO, Tufts University — Swimming cells often live in fluid environments characterized by spatial gradients of rheological properties, including biofilms and mucus layers. However, our understanding of cell transport in such environments is lacking. In this work, we use microfluidic devices to generate a spatial concentration gradient of a Newtonian polymer suspension — thus creating a viscosity gradient. Video microscopy is used to quantify the viscosity landscape and the cell motility. We demonstrate experimentally that swimming biflagellates (wild-type Chlamydomonas reinhardtii) accumulate in high viscosity regions (viscotactic response), stemming from a local reduction in cell swimming speed. A statistical analysis of the cell motility reveals that the viscous slowdown of the microalgae is due to their approximately constant flagellar thrust force in different ambient viscosities. We further demonstrate that this local viscous slowing of cell motility, leading to accumulation, is generalized in highly nonlinear viscosity gradients.

Saturday, November 23, 2019 3:00PM - 4:31PM

Session A31 Biological Fluid Dynamics: General I

3:00PM A31.00001 Machine learning in cardiovascular flows modeling: Predicting pulse wave propagation from non-invasive clinical measurements using physics-informed deep learning

- GEORGIOS KISSAS, YIBO YANG, EILEEN HWUANG, WALTER WITSCHHEY, JOHN DETRE, PARIS PERDIKARIS, University of Pennsylvania, CORLAB TEAM, PREDICTIVE INTELLIGENCE LAB TEAM — Advances in computational science offer a principled pipeline for predictive modeling of cardiovascular flows and aspire to provide a valuable tool for monitoring, diagnostics & surgical planning. Such models can nowadays be deployed on large patient-specific topologies of systemic arterial networks; but their success heavily relies on tedious pre-processing and calibration procedures that typically induce a significant computational cost. In this work we put forth a physics-informed learning framework that enables the seamless synthesis of non-invasive in-vivo measurement techniques & computational fluid dynamics models derived from first physical principles. We illustrate this new paradigm by showing how one-dimensional models of pulsatile flow can be used to constrain the output of deep neural networks such that their predictions satisfy the conservation of mass and momentum principles. Once trained on noisy and scattered clinical data of flow and wall displacement, these networks can return physically consistent predictions for velocity, pressure and wall displacement pulse wave propagation, and calculate the Windkessel model parameters as a post-processing step. 

1 Acknowledgements to: DOE(grant DE-SC0019116), DARPA(grant HR0011890034). Eunice Kennedy Shriver N’tal Institute of Child Health & Human Development (U01-HD087180), National Institute of Biomedical Imaging & Bioengineering (T32-EB009384) & NSF (DGE-1321851)

3:13PM A31.00002 A Simplified Model for the Motion of the Brain Matter in Response to Translational Impacts to the Head

- JI LANG, QIANHONG WU, Villanova University — A simplified mathematical model is developed to simulate the motion of the brain matter as our head is exposed to sudden external impacts. During the process, the cerebrospinal fluid in the subarachnoid space behaves as a squeeze damper to provide protection to the brain matter. The problem involves a solid object bathed in a liquid environment and enclosed in a container. The relative motion of the object to the container causes a squeezing flow on one side of the object and a reverse squeezing flow on the other side. A simplified theoretical model is developed in which both the inner object and the outer container are modeled as cylinders of different sizes. A constant acceleration is imposed on the container. The pressurization of the fluid and the resultant motion of the inner object is analytically predicted and validated using Ansys CFX. The result shows that the relative motion of the inner object is very sensitive to the gap thickness and the density difference between the object and the surrounding liquid. The squeezing damping effect can effectively prevent the direct contact between the solid object and the container.

1 National Science Foundation CBET Fluid Dynamics Program under Award No. 1511096
3:26PM A31.00003 Kinetics of Lactoferrin-Mediated Iron Transport through Blood-Brain Barrier Endothelial Cells. 1, PRASHANTA DUTTA, AMINUL KHAN, JIN LIU, Washington State University — Abnormally high levels of iron have been confirmed in the brain cells during several neuro-diseases such as Parkinson’s, Alzheimer’s etc. Although transferrin-mediated transcytosis is the leading mechanism for iron transport to brain, it has been found that the excessive brain irons are not resulted from transferrin. Instead, lactoferrin (Lf) is a prime suspect because of its abundant availability in the affected regions of brain. However, the kinetics of Lf-mediated iron transport is still unknown. In this paper, a mass-action based kinetic model has been presented to address the transport of Lf-mediated irons through the blood-brain barrier. The kinetic rate parameters of the model are estimated by Bayesian inference method. The robustness of the model is verified by perturbing the estimated parameters. Our results show that an increase in high affinity, but not low affinity, receptors results in higher lactoferrin as well as irons in the brain. The Lf contributes free as well as bound irons to the brain. The absence of a feedback loop such as iron regulatory proteins allows continuous transport of Lf and iron through Lf-mediated pathway, which might raise brain irons and contribute to the neurodegeneration.

1This work was supported by National Institute of General Medical Sciences of NIH under R01GM122081.

3:39PM A31.00004 Microfluidics modeling of interstitial flow through a tumor, BENJAMIN TAY, MARCOS MARCOS, Nanyang Technological University, DAVID GONZALEZ-RODRIGUEZ, Université de Lorraine — 90% of cancer-related mortalities are due to metastasis. Interstitial pressure and flow within tumor microenvironment have been identified to influence biological and biochemical aspects of metastasis. However, little is known on the biophysical effect. This study focuses on the effect of interstitial pressure and flow on the detachment of tumor cells from primary tumor. Interstitial flow through a tumor generates tensile force in the direction of flow which might contribute to detachment of tumor cells from primary tumor. Here, flow is introduced into a microfluidic device to measure pressure difference across MCF-7 aggregates with different adhesions. Aggregates with different adhesions mimic in-vivo heterogeneous tumor. Initially, fluid seeps through the porous aggregate. As pressure builds up at the afferent end, pressure-driven flow through the aggregate increases. At critical pressure difference, fracture occurs and hydraulic permeability increases. Cell masses dislodge from aggregate. Fractures occur at different critical pressure for aggregates with different adhesions. This mimic the detachment of tumor cells from primary tumor. These findings will provide better understanding about the biophysical effects of interstitial pressure and flow.

The author’s full name is Marcos.

3:52PM A31.00005 Fluid exchange dynamics during respiratory-type flows, ERIN CONNOR, AARON TRUE, MELANIE HOLLAND, JOHN CRIMALDI, University of Colorado, Boulder — Respiratory-type flows occur across a large range of scales in natural and engineered environments. These flows, characterized by the cyclic inhale and exhale of a fixed fluid volume through an orifice, result in dynamic spatiotemporal flow interactions even in quiescent surroundings. We use numerical and experimental approaches to investigate simple respiratory-type flows at scales relevant to biological sensory and metabolic processes. Organisms successfully gain access to scalars in the fluid when the amount of exhaled fluid that is subsequently re-inhaled (exchange ratio, r_E) is small. This study focuses on this exchange of fluid, whereas studies of other similar flows with ‘net-zero mass flux’ (e.g., synthetic jets) often emphasize the transport of momentum. Using time-resolved flow fields to map the Lagrangian histories of inhale and exhale cycles, we demonstrate that r_E is sensitive to small changes in Reynolds number (Re). We show that this sensitivity is due to asymmetries in the inhale and exhale flow structures; these asymmetries vanish only in the limit as Re approaches zero. We also demonstrate that r_E is sensitive to the I:E ratio, defined as the ratio of inhalation time to exhalation time. Our results suggest that organisms could optimize the exchange of fluids with their environment by modulating Re or I:E ratio.

4:05PM A31.00006 Starting jet formation through eversion of flexible plates, CHEOLGYUN JUNG, DAEGYOUM KIM, KAIST — A starting jet found in nature, such as flow through heart valves or jet propulsion of aquatic animals, is generated by the interaction of pressurized fluid with deformable membranes. The starting jet can also be formed by the interaction of pressurized fluid with deformable membranes. For example, the nematocysts of the Cnidarian animals, is generated by the interaction of pressurized fluid with deformable membranes. The starting jet flow through eversion process using a simplified model that represents an everted structure. The ends of two everted flexible plates with a large aspect ratio are clamped at both sides of a rectangular channel, and the other ends of them are in contact with each other in the middle of the channel to model the everted structure. With this configuration, we study the eversion dynamics of the flexible plates and the formation of jet flow at the exit of the channel.

4:18PM A31.00007 Acceleration-induced water ejection in the human ear canal, ANUJ BASKOTA, SEUNGHO KIM, Cornell University, HOSUNG KANG, Virginia Tech, SUNGHWAN JUNG, Cornell University — Water entering the ear canal is a common problem during swimming, showering or other water sports. The trapped water can lead to an ear infection as well as damage to the ear canal. A common strategy for emptying water from the ear canal is to shake the head, where the force created due to the head jerk helps push the water out. One-end closed hydrophobic glass tubes of varying diameters were used as a simplified model of the ear canal. Then, the tube is dropped onto a spring to mimic the shaking strategy. Results revealed that the critical acceleration to remove the water from the ear canal strongly depends on the volume and the position of trapped liquid inside the tube. We found that the critical acceleration is on the order of 10 g, which may cause serious damage to the human brain. The critical acceleration tends to be much higher in smaller sized tubes which indicates that shaking heads for water removal can be more laborious to children due to their small size of the ear canal, compared to adults.
3:00PM A32.00001 Boundary integral simulations of a red blood cell squeezing through a submicron slit under prescribed inlet and outlet pressures. ZHANGLI PENG, HUIJIE LU, University of Notre Dame, ALEXIS MOREAU, EMMANUELLE HELFER, ANNE CHARRIER, ANNIE VIALLAT, CNRS Aix Marseille University, France — We developed a boundary integral formulation to simulate a red blood cell (RBC) squeezing through a submicron slit under prescribed inlet and outlet pressures. The main application of this computational study is to investigate splenic filtrations of RBCs and the corresponding in vitro mimicking microfluidic devices, during which RBCs regularly pass through inter-endothelial slits with a width less than 1.0 micrometer. The diseased and old RBCs are damaged or destroyed in this mechanical filtration process. We first derived the boundary integral equations of a RBC immersed in a confined domain with prescribed inlet and outlet pressures. A multiscale model is applied to calculate forces from the RBC membrane, and it is coupled to boundary integral equations to simulate the fluid-structure interaction. After multi-step verifications and validations against analytical and experimental results, we systematically investigated the effects of pressure drop, volume-to-surface-area ratio, internal viscosity, and membrane stiffness on RBC deformation and internal stress. We found that spectrins of RBCs could be stretched by more than 2.5 times under high hydrodynamic pressure and that the bilayer tension could be more than 500 pN/um, which might be large enough to open mecanosensitive channels but too small to rupture the bilayer. On the other hand, we found that the bilayer-cytoskeletal dissociation stress is too low to induce bilayer vesiculation. This project is partially supported by NSF Grant CBET-1706436.

3:13PM A32.00002 Fluid-Structure Interaction of Bioluminescence. MAZIYAR JALAA, NICO SCHRAMMA, RAYMOND GOLDSSTEIN, DAMTP, University of Cambridge — Bioluminescence (emission of light from living organisms) is a common form of communication in the ocean. Here we study the bioluminescence on a single-cell level, aiming to understand the response to mechanical stimulation. In our experiments, the cell (an alga) was immobilised via micro-pipette aspiration. We impose pressure on the cell, via a submerged impinging jet. We show that the flow-induced stress on the cell membrane results in a local elastic deformation. As a result, a series of chemical signaling events occur that eventually yields to light production in sub-cellular compartments. Besides experiments, we propose a counterpart model.

3:26PM A32.00003 Flow-Induced Deformation of Cells During Small Opening Traversal. IGOR V. PIVKIN, Institute of Computational Science, USI Lugano, Switzerland — We performed experimental and computational study of cells in microfluidics measuring cell traversal through the series of openings of various sizes. MCF-10A, MCF-7 and MDA-MB-231 cells were used in experiments and corresponding computational models were developed using particle-based approach. Deformability of cells under the flow conditions will be discussed. This work was done in collaboration with the group of Chwee Teck Lim from National University of Singapore.

3:39PM A32.00004 Deformation and viability of an encapsulated cell through a microfluidic contraction. MOHAMMAD NOORANIDOOST, RANGANATHAN KUMAR, University of Central Florida — Deformation and viability of an encapsulated cell moving through a sudden contraction in a capillary tube is studied using a front-tracking method. A cell-laden droplet is initiated in a capillary tube which is allowed to migrate with the flow. When it moves through a sudden contraction, high shear stresses are experienced around the droplet where the velocity is maximum. The interplay between these stresses and interfacial forces, as well as the geometrical constraint squeezes the droplet resulting in the deformation of the inner cell. A cell viability model is used to relate the cell deformation to cell viability. Deformation and viability of the cell are highly dependent on encapsulating droplet properties and the geometry of the contraction. For a fixed geometry of the contraction, viscosity and size of the encapsulating droplet can be adjusted to minimize cell deformation. Increasing droplet size for low viscosity of the droplet shell helps reduce the deformation and maintain the viability of the cells. The deformation is enhanced for capillary tubes with narrow and long contractions, which leads to a lower cell viability. This study can be useful in biomedical applications to improve viability of cells migrating in channels with sudden obstacles.

3:52PM A32.00005 Effect of constitutive laws on erythrocyte membrane response. MARIANNA PEPONA, Duke University, JOHN GOUNLEY, Oak Ridge National Laboratory, AMANDA RANDLES, Duke University — Despite the extensive literature on the modelling of red blood cells, only a few works have compared the response of a deforming red blood cell to different constitutive laws. In the current work, three different constitutive equations are considered: the strain-hardening Skalaks law, the strain-softening neo-Hookean model, and Yeohs law, whose nature and degree of strain-hardening/softening depend on the deformation regime and type. The performance of these laws is assessed on accurately capturing deformations in the longitudinal and transverse directions, and under shear via optical tweezers, micropipette aspiration and wheel numerical experiments, respectively. Particular emphasis is given to the nonlinear deformation regime, i.e. moderate and large deformations, where it is known that the discrepancies between various constitutive laws are most prominent. Finally, we compare the aforementioned laws in the configuration of a single red blood cell flowing inside a capillary. This work aims at providing criteria for selecting the constitutive law best describing the erythrocyte membrane mechanics for the deformation type and regime of interest.

4:05PM A32.00006 Shake, rattle, and roll: microstreaming flows from acoustically oscillating cells. SCOTT TSAI, ALINAGHI SALARI, SILA APPAK-BASKOY, MICHAEL KOLIOS, Ryerson University — Steady-state microstreaming flows can arise from the interaction of sound waves with elastic objects, such as bubbles. Such microstreaming finds utility in cell and particle manipulation, and in microfluidic pumping and analyte mixing. Here, we describe a new observation that in vitro single cells that are excited by a controlled acoustic wave are also able to generate microstreaming flows. Specifically, adherent cells under the influence of a surface acoustic wave oscillate inside a microfluidic channel to generate microstreaming flows. We study the cellular properties that affect the degree of microstreaming by, for example, imposing an osmotic shock to the cell, manipulating the cell structure enzymatically using trypsin, and chemically, using paraformaldehyde and Cytochalasin D. Our findings suggest that the microstreaming induced by MDA-MB-231 cells is primarily controlled by the overall cell stiffness. We thus conclude that measuring the resulting flow pattern and velocity magnitude may be utilized as a label-free proxy for quantifying the mechanical properties, such as stiffness, of the cell.

This work is supported by the Canadian Natural Sciences and Engineering Research Council (NSERC; No. STGP 506318 & RGPIN-2017-06496). Equipment funding is from Ryerson University, the Canada Foundation for Innovation (CFI), and the Ontario Research Fund (ORF).
4:18PM A32.00007 A Microfluidic Platform for the Study of Cell Deformability

AMIR SAADAT, DIEGO A. HUYKE, INGRID H. OVREEIDE, DIEGO I. OYARZUN, Stanford University, PAULINA V. ESCOBAR, Pontificia Universidad Católica de Chile, JUAN G. SANTIAGO, ERIC S. G. SHAQFEH, Stanford University, SHAQFEH TEAM, SANTIAGO TEAM — Reduced deformability of red blood cells (RBCs) can affect the hemodynamics in the microcirculation and reduce the oxygen transport efficiency. To this end, we developed a high-fidelity computational model of RBCs in confined microchannels to inform design decisions and fabricate a microfluidic device to measure RBC deformability. We applied our computational simulation platform to determine the appropriate deformability figure(s) of merit to quantify RBC stiffness based on an experimentally measured, steady cell shape. In particular, we determined a shape parameter based on the moment of area that is sensitive to the changes in the membrane stiffness and size. We conducted experiments and developed automatic image processing codes to track the velocity and morphology of individual RBCs within microchannels. For this purpose, we fabricated PDMS microchannels with square cross-sections (7x7 μm<sup>2</sup>) and applied a small (order 10 kPa) gauge pressure at the inlet to induce cell movement (order 10 μm s<sup>−1</sup>). Our experimental setup can record 200 cells per second, and achieve image exposure times on the order of 10 μs. This microfluidic device and supporting computational tools are intended to diagnose blood cell disorders in chronic fatigue syndrome (CFS) patients relative to the healthy controls.

1Open Medicine Foundation (OMF)

Saturday, November 23, 2019 3:00PM - 4:31PM –
Session A33 Flow Instability: Richtmyer-Meshkov

3:00PM A33.00001 Characterization of Rapid Solid-Particle Dispersal by a Blast Wave

BERTRAND ROLLIN, Embry-Riddle Aeronautical University, RAHUL KONERU, BRADFORD DURANT, FREDERICK OUELLET, University of Florida — The impulsive dispersal of a bed of solid particles is often accompanied by the late time formation of coherent aerodynamic structures identified as particle jets. Despite numerous experimental and numerical studies to date, the intricate mechanisms leading to the formation and selection of these jets have not been conclusively characterized. In light of recently published experimental work, numerical simulations of the performed using the quasi-two-dimensional geometric setting of a Hele-Shaw cell. Explicitly, a dense and uniform volumetric distribution of solid particles shaped in the form of a right circular hollow cylinder is sandwiched between two solid plates separated by a small distance. The initial impulse to the particles is then given by a relatively weak air blast wave. The highly resolved point-particle simulations focus on the interplay between particle inertia and the gas-solid particle limit of classic hydrodynamic instabilities such as the Rayleigh-Taylor and the Richtmyer-Meshkov instabilities, in explaining the formation of internal and external particle jets. The initial overpressure and particle properties are used as control parameters in the quantification of the characteristics of the late-time particle jets.

1This work was supported by the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, Contract No. DE-NA0002378.

3:13PM A33.00002 Attenuation of transmitted shock by a particle-seeded layer

PETER VOROBIEFF, University of New Mexico, GUSTAAF JACOBS, TZ-TING HUNG, San Diego State University — We conduct an experimental and numerical study of a nominally planar interaction of a normal shock with a layer of particles embedded in gas (air). Prior experimental studies reveal that even a particle layer with a modest volume fraction of particles (1-9%) produces a reflected pressure wave at Mach numbers above 1.4. The present work focuses on two aspects of the flow. First, we examine the transmitted pressure wave, using both experimental measurements and computational modeling, and seek to identify the influence of the system used to form the particle layer in experiments and any effects peculiar to the layer thickness. Second, we present a simple model describing perturbation growth in the gravity-driven particle layer before the shock arrival.

1This research is supported by NSF grant 1603915.

3:26PM A33.00003 Acceleration of a Vortex Ring-Deformed Stratified Interface by a Planar Shock Wave

ALEX AMES, UW-Madison, CHRIS WEBER, Lawrence Livermore National Laboratory — Vortices are a well-known mechanism of transport across stratified interfaces. When repeatedly shocked, preexisting vorticity-driven deformation of the interface provides greater leverage for baroclinic torque under subsequent acceleration. The locally axisymmetric baroclinic vorticity deposition is a primary source in the turbulent energy cascade, leading to anisotropically-oriented eddies at the inertial scale, producing deleterious interfacial mixing. In particular, the assembly of a uniform fusion hotspot in inertial confinement fusion capsules is disrupted by the protrusion of a cold vortical bubble arising from the remains of the fill tube. The morphology and evolution of compressible, variable-density vortices upon shock acceleration is explored computationally using the MIRANDA hydrodynamics code. A laboratory-scale configuration comprising a vortex ring discharged upwards into a stably-stratified layer from the open end of a small shock tube is compared for hydrodynamic similarity to a representative ICF fill tube perturbation. Vortex evolution, baroclinic production during and following shock interaction, behavior of secondary vortices, and mixing intensification are detailed across a range of vortex strengths and Atwood & Mach numbers.

1This work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344

3:39PM A33.00004 The effect of membraneless initial conditions on the growth of Richtmyer-Meshkov instability.

MOHAMMAD MANSOOR, SEAN DALTON, ADAM MARTINEZ, TIFFANY DESJARDINS, JOHN CHARONKO, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — The Richtmyer-Meshkov Instability (RMI) is described by the baroclinic generation of vorticity at a density stratified interface when impulsively accelerated. Here, we experimentally investigate the late-time RMI growth of sinusuous perturbations of an air/sulfur hexafluoride interface subjected to a Mach 1.2 planar shock wave within the vertical shock tube (VST) facility at Los Alamos National Laboratory. Interface perturbations are established using a novel membraneless technique where cross-flowing Air and SF6 separated by oscillating splitter plate enter the shock tube with an undulating structure. It is found that late-time perturbation growth behavior depends significantly on initial perturbation wavelength and peak-to-valley amplitude as prescribed by the frequency and sweeping angles of the “oscillating plate”. The results are compared with past nonlinear models for various scaled initial amplitudes (ka<sub>0</sub>) and used to propose an empirical rational function that captures the asymptotic behavior of perturbation growth for both low and high scaled initial amplitudes.
3:52PM A33.00005 Time-Resolved Particle Image Velocimetry of the 3-D, Multi-Mode Richtmyer-Meshkov Instability. JEFFREY JACOBS, EVEREST SEWELL, KEVIN FERGUSON. The University of Arizona — We present recent experiments conducted on the multi-mode Richtmyer-Meshkov instability (RMI) using time-resolved Particle Image Velocimetry (PIV), comparing results initiated using high and low amplitude initial perturbations. Measurements of the growth parameter $\theta$ indicate a slight difference in growth rate exists between the two groups following the incident shock interaction, with additional differences in the growth of the instability following reshock. We validate a novel method of obtaining $\theta$ from the decay of turbulent kinetic energy (Thorburn et al., J. Fluid Mech., 2010). Examination of the anisotropy ratio reveals an asymptotic value of approximately 1.8 in high amplitude experiments, while low amplitude experiments exhibit decreasing anisotropy. High amplitude experiments exhibit an inertial range, while low amplitude experiments remain below the threshold for inertial range for most times. An extension of this analysis for developing flows (Zhou et al., Phys Rev E, 2003) reveals that neither flow satisfies the extended transition (Dimotakis, J. Fluid Mech., 2000) following the incident shock, with low amplitude experiments remaining below the threshold for mixing transition. This observation is supported by the lack of an apparent inertial range in the power spectra of velocity.

4:05PM A33.00006 Scalar Power spectrum and Structure Function analysis of the Richtmyer-Meshkov Instability Upon Re-Shock$^1$. CHRISTOPHER NOBLE, JOSHUA HERZOG, ALEX AMES, JASON OAKLEY, DAVID ROTHAMER, RICCARDO BONAZZA, University of Wisconsin - Madison — The Richtmyer-Meshkov instability of a twice-shocked gas interface is investigated in the vertical shock tube of the Wisconsin Shock Tube Laboratory at the University of Wisconsin–Madison. The initial condition is a shear layer, containing broadband perturbations, formed at the interface between a helium-acetone mixture and argon. The interface is accelerated with a shock of nominal strength $M=1.9$ with an initial Atwood number of $A=0.43$. Acetone is used as a molecular tracer for PLIF, allowing the extraction of concentration data by using a pulse burst laser system at 20kHz to excite acetone to fluoresce. The resulting fluorescence signal is measured using a high-speed Phantom camera. The evolution of the scalar power spectrum is investigated. As seen in previous single shock experiments a region of -5/3 slope is seen at late post-shock times, however at late re-shock times a larger region of -8/3 spectrum is observed. The measurement limit of the present experiments is estimated to be within the inertial range that may exist thus the measured slope is not expected to be a dissipation effect but the slope of the inertial range. Scalar structure functions are calculated, with the anomalous exponent being plotted against the structure function order also showing a non-KOC scaling. The terms in the scalar power spectrum evolution equation are calculated showing an asymmetry about the centre of the mixing layer and suggesting the emergence of an inertial range.

1U.S. DOE/NNSA grant number DE-NA0002935

4:18PM A33.00007 High-resolution, time-resolved PIV measurements on the Richtmyer-Meshkov instability in a dual-driver vertical shock tube. KEVIN FERGUSON, EVEREST SEWELL, JEFFREY JACOBS, The University of Arizona — Experiments on the Richtmyer-Meshkov Instability (RMI) using Particle Image Velocimetry (PIV) in a dual driver vertical shock tube are presented. Two shock waves generated at opposite ends of a vertical shock tube travel in opposing directions, impacting a perturbed interface formed between Air and Sulfur Hexafluoride ($\text{SF}_6$). Perturbations are formed using a pair of voice coil driven pistons that generate Faraday waves on the interface. The incident shock wave arrives from the air side of the interface which initiates the RMI. Shortly afterward a second shock wave arrives from the $\text{SF}_6$ side which generates reshock. Shock strengths are chosen to result in halfed interface motion after passage of the second shock wave, permitting a long observational window in which the instability can evolve and yielding a simplified optical and recording setup as compared to typical single-driver experiments. Four cameras are utilized in a tiled pattern to create a high-speed recording of each experiment with a greatly increased final vector resolution compared to previous experiments. Information on the growth of the RMI, including measurements of the growth exponent, $\theta$, anisotropy, and turbulent kinetic energy decay are presented.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A34 Flow Instability: Transition to Turbulence 616 - Jae Sung Park, UNL

3:00PM A34.00001 Transition in Pulsatile Flows with Flow Reversals$^1$. JOAN GOMEZ, YIANNIS ANDREPOULOS, City College of New York, YIANNIS ANDREPOULOS TEAM — Pulsatile flows are of interest because for certain range of Reynolds and Womersley numbers they exhibit flow reversals while at the same time they display laminar and turbulent behavior at different times of the pulsating cycle. Addressing how turbulence appears, decays and is suppressed in such environments is challenging due to the flow unsteadiness and flow-wall interactions. An experiment was setup to replicate pulsatile motion of water flowing in a clear, rigid pipe. The flow is driven by a piston-motor assembly controlled by a computer to induce cyclic motion of the mean flow. Time-Resolved Particle Image Velocimetry (TR-PIV) techniques are used to acquire velocity data on the plane of a CW laser illumination sheet. Simultaneous acquisition of time-dependent PIV data and wall pressure measurements, obtained from pressure sensors installed along the length of the pipe, allow the estimation of the instantaneous wall shear-stress which was used as a metric for the appearance of reverse and forward flow regions. It was found that the reverse flow region is formed close to the wall and it is bounded from the forward flow region by a counter-flowing shear-layer. Transition to turbulence occurs within this shear layer which prevents propagation of disturbances from the near wall region towards the center of the pipe.

1Fundied by NSF grant: CBET 1803845

3:13PM A34.00002 The impact of slip surfaces on exact coherent states: insight into the transition-to-turbulence$^1$. ETHAN DAVIS, JAE SUNG PARK, University of Nebraska-Lincoln — The effect of slip surfaces on the laminar-turbulent separatrix, or transition, in a channel geometry is studied by direct numerical simulation. Firstly, turbulence lifetimes, or the likelihood that turbulence persists, is investigated. Slip surfaces decrease the likelihood of sustained turbulence. The likelihood is decreased further with increasing slip length. Secondly, a more deterministic approach is used to investigate the effect of slip surfaces on the transition to turbulence. To this end, exact coherent states, specifically nonlinear traveling wave solutions to the Navier-Stokes equations, are used as initial conditions. Lower-branch solutions are insightful for the study of transition as they lie along the laminar-turbulent boundary in state-space. Two solution families, P3 (core mode) and P4 (critical layer mode), are considered. For P3, slip surfaces delay the transition, whereas for P4, viscosity is the primary mechanism. In general, slip surfaces delay the transition while weakening the instability. Beyond a critical slip length, the instability is totally eliminated, and flow is laminarized. Flow dynamics and structures are further discussed for this transition events.

1This work was supported in part by the National Science Foundation, Grant No. OIA-1832976.

3:26PM A34.00003 ABSTRACT WITHDRAWN —
of wild type cells and a mutant lacking flagella, suggesting that CO

Cholerae 1800 (Schlichting jet) or use 2D perturbations, which do not allow for spanwise variation (Bickley jet). The 3D normal modes are then investigated of the Schlichting jet and its planar counterpart, the Bickley jet, most of which, to our knowledge, either require perturbations to be axisymmetric three-dimensional (3D) normal-mode perturbations to the 2D base flow. We compare our results to previously published linear stability analyses of the Schlichting jet and its planar counterpart, the Bickley jet, most of which, to our knowledge, either require perturbations to be axisymmetric or use 2D perturbations, which do not allow for spanwise variation (Bickley jet). The 3D normal modes are then investigated using DNS to understand the nonlinear saturation (in the sense of growth and maximization of perturbation energy) and the transition of the flow to a fully turbulent state.

Spatial amplification of coupled disturbances in bluff body shear layers, DANIEL MOORE, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Detailed experimental campaigns into separated shear layers have been carried out on a series of rectangular sections over a Reynolds number range from 13,400 to 118,000, based on the body height, at a range of angles of attack. Recent work by this same group has demonstrated the level of coupling that occurs between the convective shear layer instability at the leading and the global, large-scale instability associated with wake shedding of these sharp-edged rectangular sections. Building upon these findings, the spatial amplification of the separated shear layers is experimentally derived and compared to other shear flows including the classical planar mixing layer. Results show that the effect of a coupled shear layer manifests itself via a relatively broad spectrum of unstable frequencies and significantly elevated growth rates. On the other hand, reducing the coupling the same two instabilities has the opposite effect. It is further shown that the coupling is directly tied to a turbulent transition length, reinforcing the notion of a globally unstable flow field.

Temporal development of hairpin vortex sequences in turbulent puffs, KYLE WINTERS, ELLEN LONGMIRE, University of Minnesota, DEPARTMENT OF AEROSPACE ENGINEERING AND MECHANICS TEAM — Puffs are characterized by intermittent swirling structures that develop from sufficient perturbations to pipe flow around 1800 < Re < 7000. By conducting stereo-PIV on circular cross sections, the authors examined hairpin-shaped vortices near the trailing edge of many puffs. To study the temporal evolution of the hairpins, planar-PIV was conducted on an axial-wall normal plane inside a 44.8 mm diameter, D, 8.8m (180D) downstream of a disturbance ring. In certain records, the measurement plane coincided with the plane of symmetry of a hairpin. This allowed for comparison with the stereo-PIV records and examination of the hairpin development. The hairpin heads and associated velocity fluctuations grew in strength as they moved both downstream and away from the wall. Eventually the original hairpin spawned a new hairpin upstream close to the wall at the same azimuthal position. The axial spacing between the two hairpins agreed with that observed in the stereo-PIV records. Further, the spacing grew as they continued to propagate downstream at velocities that generally agree with the propagation of axial fluctuations noted by Shimizu and Kida (2008). The overall development process was similar to that outlined by Jodai and Elsinga (2016) in a turbulent boundary layer.

The onset of turbulence in channel flow, BJORN HOF, VASUDEVAN MUKUND, CHAITANYA PARANJAPE, PHILIP SITTE, Institute of Science and Technology Austria — In channel flow turbulence arises at Reynolds numbers where the laminar state is linearly stable and in order to do so the critical point where turbulence first becomes sustained we identify the growth and decay processes of individual turbulent stripes in experiments. As shown, the critical point is reached when the (continuous) entrainment of laminar fluid at the stripes downstream tip outweighs the (stochastic) shedding of turbulence at their upstream tip. For growing stripes, the probability to collapse decreases while the probability to split increases in time. Consequently, neither collapse nor splitting are memoryless and unlike in pipes and unlike in directed percolation, the contamination rate changes in time. The coupling between the growth of individual stripes and the creation of new stripes leads to a significantly lower critical point than most earlier studies suggest.

Saturday, November 23, 2019 3:00PM - 4:31PM — Session A35 Electrokinetic Flows: General 617 - Michael Booty, NJIT

CO₂-driven diffusiophoresis: motion of charged particles near CO₂ dissolving boundaries, SUIN SHIM, Princeton University, OREST SHARDT, University of Limerick, SEPIDEH KHO-DAPARAST, Imperial College London, CHING-YAO LAI, Columbia University, JESSE T. AULT, Brown University, BHARGAV RALLABANDI, University of California, Riverside, HOWARD A. STONE, Princeton University — We present experimental and theoretical investigations for CO₂-driven diffusiophoresis of charged particles. An aqueous suspension of charged particles initially in contact with a CO₂ dissolving interface shows directional migration either up or down the concentration gradient, depending on the particle charges. We use a cylindrical CO₂ bubble in a circular Hele-Shaw cell to study the behavior of polystyrene particles. By experiments and numerical model calculations considering multicomponent gas dissolution, we prove the diffusiophoretic accumulation and exclusion of particles near the bubble interface. Then using two geometrical conditions with moving and fixed boundaries, we show diffusiophoresis of bacterial cells driven by dissolution of CO₂. Vibrio Cholerae, a Gram-negative bacteria, has a negative surface charge and thus migrates away from a CO₂ source. Using PIV we show behaviors of wild type cells and a mutant lacking flagella, suggesting that CO₂-driven diffusiophoresis may prevent biofilm formation by reducing the population of cells approaching an interface.

We acknowledge the NSF for support via grant CBET-1702693.

AC Electrokinetics in the Limits of Extremely High Voltage and Ion-Asymmetry, ARUNRAJ BALAJI, SHAHAB MRJALILI, ALI MANI, Stanford University — The aim of this study is to extend understanding of AC electrokinetics to regimes involving Dielectric Barrier Discharge (DBD) plasmas. To this end, we consider a simple model utilizing the Poisson-Navier-Planck and Navier-Stokes equations, as commonly used in analysis of electrokinetic phenomena. To study the specific regimes relevant to plasmas, we consider the limits of extremely high AC voltage (\( V = O(10^4 V_o) \)) and extremely high ion-asymmetry (\( \frac{\epsilon_{\text{ion}}}{\epsilon_{\text{water}}} = O(10^4) \)). Direct numerical simulation (DNS) of the governing equations is performed in one- and two-dimensional spatial domains, and different regimes of response are identified in terms of input voltage and diffusion asymmetry parameters. In addition to the results of DNS, this work also focuses on modeling considerations for the plasma regime and steps toward a more complete simulation of DBD: one that includes the effects of thermal-nonequilibrium and reactions.

Supported by the Precourt Institute for Energy
3:26PM A35.00003 A model for electrokinetic flow with deformable interfaces, MICHAEL BOOTY, New Jersey Institute of Technology, MANMAN MA, Tongji University, Shanghai, MICHAEL SIEGEL, New Jersey Institute of Technology — A hybrid or multiscale model is introduced to describe the evolution of a drop in the two-phase flow of immiscible, ionic fluid electrolytes that are driven by an electric field. It is formed from the PNP equations in the Stokes flow regime in the limit when the Debye layers are thin relative to the undeformed drop size. For arbitrary deformation, the model consists of boundary integral equations for the electrostatic potential and the interface fluid velocity together with relations that contain the coupling between the electrostatic and fluid fields within the thin Debye layers. The results of sample numerical simulations are presented, together with comparison to a small-deformation analysis in the limit of a weak applied field; further generalizations of the model are also discussed.

3:39PM A35.00004 Characterization of surface-solute interactions by diffusioosmosis, JESSE AULT, Brown University, SANGWOO SHIN, University of Hawaii at Manoa, HOWARD STONE, Princeton University — The measurement of wall zeta potentials and solute-surface interaction length scales for electrolyte and non-electrolyte solutes, respectively, is critical to the design of biomedical and microfluidic applications. We present a microfluidic approach using diffusioosmosis for measuring either the zeta potentials or characteristic interaction length scales for surfaces exposed to, respectively, electrolyte or non-electrolyte solutes. When flows containing different solute concentrations merge in a junction, local solute concentration gradients can drive diffusioosmosis due to interactions between the solute molecules and solid surfaces. We demonstrate a microfluidic system in which solute concentration gradients drive diffusioosmosis within a pore, resulting in predictable fluid and solute profiles. Furthermore, we present analytical results and a methodology to determine the zeta potential or interaction length scale for the pore surfaces in the system. We apply this method to the experimental data of Lee et al., and we use 3D numerical simulations to validate the theory. To the best of our knowledge this is the first flow-based approach to characterize surface/solute interactions with non-electrolyte solutes.

3:52PM A35.00005 ABSTRACT WITHDRAWN

4:05PM A35.00006 Microscale Electrodeionization: in situ Concentration Profiling and Flow Visualization, SUDONG PARK, RHOKYUN KWAK, Hanyang University, Korea — Electrodeionization (EDI) is a membrane-based desalination system utilizing ion exchange membranes and resins. By combining electrodialysis and ion exchanger, EDI can produce ultrapure water in continuous-flow manner. Although its theoretical mechanisms are well documented, there is no experimental platform which can provide microscopic details inside the system. In this paper, we present microscale EDI that visualizes in situ ion concentration, pH, and fluid flow. The platform was fabricated by filling ion exchange resins as a monolayer in a transparent polydimethylsiloxane channel between cation/anion exchange membranes. According to operating voltages (0-15V), distinct behaviors of ion concentration profile, pH shift, and fluid flow were observed: ohmic, limiting, and overlimiting regimes. It is noteworthy that overlimiting regimes can be sub-categorized as water-splitting regime and electroconvection regime. In early stage (4-6V), water-splitting is dominant with pH change near the membranes and resins; under higher voltage (8-15V), electroconvection start to occur even water-splitting tries to suppress the development of the electroconvective instability on the resins.

1This work was supported by the Climate Change Response Technology Development Project (NRF-2017M1A2A047475) and the Basic Research Project (NRF-2019R1C1C1008262) from the National Research Foundation of Korea.

4:18PM A35.00007 Non-isothermal electrokinetic effects in an electro-osmotic flow, EDGAR RAMOS, FEDERICO MENDEZ, National Autonomous University of Mexico, JOSE LIZARDI, College of Science and Technology — In this work, we study numerically the combined influence of some non-isothermal electrokinetics effects derived from a Debye length versus fluid viscosity, when both depend on the temperature for an electroosmotic laminar flow circulating in a slit microchannel. Considering then that the Debye length depends on temperature T, together with the fluid viscosity, we obtain additional temperature gradients along the microchannel and the isothermal hypothesis is no longer valid. Therefore, the Navier-Stokes equations together with the energy, Poisson and Ohmic current conservation equations are solved by using a routine finite element method. For this purpose, the governing equations are written in a dimensionless format and we introduce a dimensionless thermal parameter α that measures the temperature deviations of a reference temperature and a dimensionless parameter γ, which defines the effects of the variable viscosity. The numerical predictions show that the influence of these parameters yields different regimes for the behavior of the volumetric flow rate in comparison with a uniform Debye length and strong induced pressure gradients are sensibly altered by the existence of these dimensionless parameters.

Saturday, November 23, 2019 3:00PM - 4:31PM —
Session A36 Microscale Flows: Assembly and Fabrication 618 - Boris Stoerber, University of British Columbia

3:00PM A36.00001 Colloidal particle dynamics during band assembly, ANDREW YEE, MINAMI YODA, Georgia Institute of Technology — Evanescent-wave visualizations have shown that colloidal polystyrene particles in a dilute (volume fractions < 4 × 10^{-3}) suspension assemble into structures called “bands” that only exist near the walls in combined Poiseuille and electroosmotic “counterflow” through silica and polydimethylsiloxane-silica microchannels. These bands have cross-sectional dimensions of a few μm and a length comparable to that of the channel of a few cm. Two-color experiments, where ~1% of the α = 250 nm particles are labeled with a different fluorophore, are used to investigate particle dynamics over time for a range of flow conditions. In the initial accumulation stage, the near-wall particle concentration increases sharply, and continues to increase after the first band is observed, before decreasing to a roughly constant value with a stable number of “steady-state” bands. Although the particles appear to be in a liquid state within these structures, they do not follow the flow, even before the bands form, and have negligible cross-stream motion. The velocities of near-wall particles within the steady-state bands are found to be much less than those between the bands.

1Supported by US Army Research Office
3:13PM A36.00002 Solvent-phase electrodeposition of highly concentrated, vertically aligned carbon nanotubes for scalable fabrication of carbon-nanotube membranes. RICHARD CASTELLANO, Rutgers University, ERIC MESHOT, FRANCESCO FORNASIERO, Lawrence Livermore National Labs, ROBERT PRAINO, Chasm Technologies, JERRY SHAN, Rutgers University — Membranes incorporating vertically aligned carbon nanotubes (VA-CNTs) as through-pores have been shown to transport fluids at rates orders-of-magnitude faster than predicted by theory, offering promise as highly permeable membranes for applications as diverse as protective yet breathable garments, and desalination membranes. There is a need for cost-effective and scalable methods for fabricating VA-CNT membranes. Here, we describe a solution-based fabrication technique for VA-CNT membranes using electric-field alignment and electrophoretic concentration of CNTs initially dispersed in a solvent. The quality of electrodeposition is described in terms of the resulting CNT number density. Multiple rounds of solvent-phase CNT alignment and deposition are performed, before the solvent is finally replaced with a UV-curable prepolymer, all while an electric field is applied to preserve CNT alignment. The liquid prepolymer is cured to form membranes, which are shown to have open CNT pores with flow enhancement. We compare the solution-fabricated VA-CNT membranes to those fabricated by other, less-scalable methods, and discuss the implications for large-scale production of such membranes.


3:26PM A36.00003 Multiphase modeling of precipitation-induced membrane formation1, PATRICK EASTHAM, NICK MOORE, NICK COGAN, Department of Mathematics, Florida State University, QINGPU WANG, OLIVER STEINBOCK, Department of Chemistry and Biochemistry, Florida State University — We have formulated a model for the dynamic growth of a membrane developing in a flow as the result of a precipitation reaction, a situation inspired by recent microfluidic experiments. A key challenge is that the location of the immobile membrane is unknown a priori. To model this situation, we use a multiphase framework with fluid and membrane phases; the aqueous chemicals exist as scalar fields that react within the fluid to induce phase change. Analysis demonstrates no-slip behavior on the developing membrane without a priori assumptions on its location, with additional numerical simulation in 2D microfluidic geometries. The model has applications towards precipitate reactions where the precipitate greatly affects the surrounding flow, a situation appearing in many laboratory and geophysical contexts. More generally, this model can be used to address fluid-structure interaction problems that feature the dynamic generation of structures.

1 NSF (4) & Simons Collaboration (1) Grants

3:39PM A36.00004 Dynamic elastocapillarity of hairy tubes. JONGHYUN HA, KAIYING JIANG, SAMEH TAWFICK, University of Illinois at Urbana-Champaign — Elastocapillarity, which is the capillary-driven deformation of slender materials, can be mundanely observed in our daily lives, such as painting, washing the hair, wet grass or leaves. Here, we introduce a novel elastocapillary phenomenon in hairs assembled into ring-shaped cross sections thus forming hairy-tubes composed of an empty hole surrounded by a hairy wall. The heterogeneous hairy tubes have two distinct spacing length scales: the narrow spacing among the individual hairs and the large inner diameter of the tube which is a few millimeters. The hairy tubes are immersed in a liquid bath, and once they pierce the liquid interface, the fibers self-assemble due to the capillary action. In particular, we observe that the drainage dynamics between the fibers play an important role in the deformation trend, which has two distinct modes. The fibers locally coalesce in the low drainage rate forming tubes having smaller inner and outer diameters than the dry counterparts with denser fiber packing within the walls, while they completely collapse into round bundles and eliminate the internal diameter at the high drainage rate. Based on the physics of elastocapillarity, we theoretically and experimentally explain the shape shifting induced by surface tension, depending on the structure size and the drainage speed. This study provides the model system of capillary induced self-assembly of heterogeneous hairy structures, which have far more applications, such as micro/nanoscale manufacturing and soft actuators.

3:52PM A36.00005 Electric field driven aggregation of negatively and positively polarized particles in dilute suspensions1. BORIS KHUSID, QIAN LEI, EZINWA ELELE, New Jersey Institute of Technology — We will present data on the electric field driven particle aggregation in dilute suspensions of nearly neutrally-buoyant negatively and positively polarized particles. Following the application of a sufficiently strong high-frequency AC field to both suspensions, particles aggregated head-to-tail into chains that bridged the gap between two electrodes. Once a weak DC field was added, negatively polarized particles formed a cellular pattern, in which large-scale particle-free domains were enclosed by particle-rich thin walls. The appearance of such patterns in suspensions of negatively polarized particles in a strong AC field was observed by Kumar, Khusid, Acrivos, PRL95, 2005 and Agarwal, Yetihra, PRL102, 2009. However, formation of cellular structures is not predicted by current theories for the field induced particle aggregation in polarized suspensions.

1 Supported by NASA’s Physical Science Research Program, NNX13AQ53G

4:05PM A36.00006 Fabrication of water-in-water colloidosomes from aqueous two-phase systems using droplet microfluidics. WEI GUO, YAGE ZHANG, SHIPEI ZHU, ANDERSON H. C. SHUM, Department of Mechanical Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong — We describe the fabrication of stable and monodispersed colloidosomes derived from water-in-water microdroplets using an integrated high-throughput microfluidic system. Liquid-liquid phase separation of aqueous two-phase systems (ATPS) inside the microdroplets is used as the driving force to form the semipermeable shell of colloidosomes. Droplets of water-in-oil are firstly generated in a microfluidic flow-focusing device, where aqueous dextran solutions with the addition of negative-charged nanoparticles are used as dispersed phase. Then we use a pico-injector to introduce another component, aqueous polyethylene glycol (PEG) solutions with the addition of positive-charged polyelectrolytes, into the microdroplets. Phase separation inside the pico-injected droplets happens due to the non-equilibrium osmotic pressure between the two components, triggering the formation of crosslinked shells by complex coacervation. Our confocal images and stiffness test show that these shells have stable morphology and robust structures. Finally, we use a microfluidic sorting module to transfer those droplets containing crosslinked shells from oil phase to aqueous PEG phase, allowing the fabrication of water-in-water colloidosomes in a high-throughput way.
interphase coupling. We are able to estimate subgrid force fluctuations reasonably well, and thereby greatly enhance the fidelity of mesoscale simulations through improved modeling.

70% of the observed force variation. Since precise location of each particle is known in an Eulerian-Lagrangian simulation, our model would be able to correlate the microscale variation of the drag and lift force with the local anisotropy of each particle’s neighborhood. Given the distribution maps extracted from direct numerical simulations, our model takes advantage of the statistical information of the arrangement of neighbors to predict the macroscopic net particle deposition profile. Our work thus demonstrates how pore-scale transport dynamics of colloidal particles in a model 3D porous medium using confocal microscopy. For the first time, we characterize the interplay between particle deposition and flow in a complex 3D porous medium. However, this behavior is difficult to model due to diverse processes that may arise, including capillary forces and fluid flow.

Our research was executed through the NETL Research and Innovation Centers Advanced Reaction Systems FWP. Research performed by Leidos Research Support Team staff was conducted under the RSS contract 89243318CFE000003.

**3:39PM A37.00004 Colloidal Particle Flow through Porous Media: A Multiscale Study**

NAVIN BIZMARK, Princeton Institute for the Science and Technology of Materials, Princeton University

RODNEY PRIESTLEY, SUJIT DATTA, Department of Chemical and Biological Engineering, Princeton University

RODNEY PRIESTLEY, SUJIT DATTA, Department of Chemical and Biological Engineering, Princeton University — Colloidal particles hold promise for improved oil recovery and groundwater aquifer remediation. These applications rely on the transport of injected particles through a subsurface three-dimensional (3D) porous medium. However, this behavior is difficult to model due to diverse processes that may arise, including particle advection through the pore space by fluid flow, adsorption and deposition onto the solid matrix, and erosion or resuspension. Moreover, these processes are particularly difficult to study experimentally due to the opacity of typical 3D media. Here, we directly visualize the transport of colloidal particles in a model 3D porous medium using confocal microscopy. For the first time, we characterize the interplay between particle adsorption and erosion at the pore scale. Analysis of these pore-scale processes allows us to determine the net particle deposition rate, which enables us to predict the macroscopic net particle deposition profile. Our work thus demonstrates how pore-scale transport dynamics of colloidal particles can be controlled to achieve desired macroscopic goals with consequences for oil recovery, aquifer remediation, and other emerging applications.
and Waals interlayer force and leads to exfoliation. By studying two exfoliation processes with different hydrodynamics, however, it has become

in a process called liquid phase exfoliation. Shear stress is commonly understood to be the leading ‘parameter, as it overcomes the van der

MATAR, Imperial College London — Graphite particles dispersed in a solvent can be exposed to high shear stresses in order to produce graphene

Exfoliation

conditions (temperature, humidity), and the particle size distribution. The presented experimental facility is designed with the aim to maximize

flow conditions. The parameters under consideration include the flow Reynolds number, the powder loading, the particle material, the ambient

section. We quantify deposit formation through the characterization of the pattern and the thickness of the deposit layers depending on the

caused in the past numerous dust explosions. In order to contribute to operational safety of industrial plants we explore the deposit of particle-

deposit layers on component surfaces. The resulting local accumulation of electrostatic energy can lead to hazardous spark discharges which

Universit catholique de Louvain — When powders are transported pneumatically they often gain an electrostatic charge and form subsequently

1

be discussed in detail for three post geometries.

is high compared to the surface tension of the interface, the non-wetting liquid pierces through the pockets of the entrapped receding liquid

performed for various Weber numbers (We) of the invading liquid for three flow geometries. The entrapped volume was found to increase

the liquid entrapment process as the liquid-liquid interface evolves through a quasi-two-dimensional pore doublet model. Experiments are

In these situations, it is typically desired to minimize entrapment of the receding fluid in the pores. In this study, we investigate experimentally

mechanisms associated with exfoliation and determine the optimal operating points.

rotational speed of the cylinder have a significant effect on the resultant graphene production rate. Detailed experimental and CFD studies,

time the particles were exposed to the shear. In the second setup, based around Taylor-Couette flow, the influence of both the pump speed and

a spinning disc, increasing the rotational speed and/or flow rate leads to an increased radial velocity, thereby directly reducing the amount of

caus in the past numerous dust explosions. In order to contribute to operational safety of industrial plants we explore the deposit of particle-

transport1, HOLGER GROSSHANS, NUKI SUSANTI, Physikalisch-Technische Bundesanstalt (PTB), MILTIADIS V. PAPALEXANDRIS, Univer-

universit catholique de Louvain — When powders are transported pneumatically they often gain an electrostatic charge and form subsequently

1

The authors gratefully acknowledge the financial support from the Max Buchner Research Foundation.

4:05PM A37.00006 Experimental evaluation of deposit formation during powder

4:18PM A37.00007 Optimisation of Graphene Production via Liquid Phase

1We acknowledge funding from the European Union Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie

grant agreement No 707340

Saturday, November 23, 2019 3:00PM - 4:31PM –

Session A38 Porous Media Flow: Multiphase Flows

3:00PM A38.00001 Controlling capillary fingering using pore size gradients in dis-

ordered media, NANCY LU, CHRISTOPHER BROWNE, DANIEL AMCHIN, Department of Chemical and Biological Engineering, Princeton University, JANINE NUNES, Department of Mechanical and Aerospace Engineering, Princeton University, SUJIT DATTA, Depart-

dment of Chemical and Biological Engineering, Princeton University — Capillary fingering is a displacement process that can occur when a

non-wetting fluid displaces a wetting fluid from a homogeneous disordered porous medium. Here, we investigate how this process is influenced

by a pore size gradient. Using microfluidic experiments and computational pore-network models, we show that the non-wetting fluid displacement

behavior depends sensitively on the direction and the magnitude of the gradient. The fluid displacement depends on the competition between a

pore size gradient and pore-scale disorder; indeed, a sufficiently large gradient can completely suppress capillary fingering. By analyzing capillary

forces at the pore scale, we identify a non-dimensional parameter that describes the physics underlying these diverse flow behaviors. Our results

thus expand the understanding of flow in complex porous media and suggest a new way to control flow behavior via the introduction of pore

size gradients.

3:13PM A38.00002 Evolution of a liquid-liquid interface through a symmetric pore

doublet model.1, JONATHAN SILES GARNER, ANCHAL SAREEN, ELLEN LONGMIRE, University of Minnesota — In applications

such as oil recovery, diagnostics and printing, the displacement of one liquid from a porous medium is often achieved by injecting another liquid.

In these situations, it is typically desired to minimize entrapment of the receding fluid in the pores. In this study, we investigate experimentally

the liquid entrapment process as the liquid-liquid interface evolves through a quasi-two-dimensional pore doublet model. Experiments are

performed for various Weber numbers (We) of the invading liquid for three flow geometries. The entrapped volume was found to increase

significantly with increasing Weber number of the invading fluid. Beyond a ‘critical’ We, when the inertia of the invading non-wetting liquid

is high compared to the surface tension of the interface, the non-wetting liquid pieces through the pockets of the entrapped receding liquid

leading to ‘instability’. It was also found that the entrapped volume can be reduced significantly by making the trailing edge of posts pointed.

This geometry also delays the interface ‘instability’ to higher We. Effects of the fluid viscosity ratio and We number of the invading fluid will

be discussed in detail for three post geometries.

1This work is supported by the ACS Petroleum Research Fund.
3:26PM A38.00003 Numerical Simulation of Immiscible Pore Scale Flow: Wettability and Dynamics, SOHEIL ESMAEILZADEH, ZHIPENG QIN, Stanford University, AMIR RIAZ, University of Maryland, HAMDI TCHLEPI, Stanford University — Accurate characterization of fluid-fluid interfacial dynamics is crucial for modeling pore-scale multiphase flows common in water resources management and subsurface applications. In this work, we propose a framework to accurately capture the dynamics of the capillary dominated pore-scale fluid-fluid interfaces in the presence of complex-shaped confinements. The incompressible Navier-Stokes equations are coupled with a multiscale sharp-interface level-set method and a direct-forcing based immersed boundary approach on a cartesian mesh to capture the interfacial dynamics. With the viscous terms being treated semi-implicitly, and a dynamic contact-line model suited for curved surfaces, we study the effects of wettability, contact angles, and pinch-off dynamics.

Keywords: water resources management, pore-scale, level-set method, immersed boundary, multiphase flow

3:39PM A38.00004 Anisotropic wicking, SOHYUN JUNG, Seoul National University, WONJUNG KIM, Sogang University, HO-YOUNG KIM, Seoul National University — Capillarity-driven wicking of liquids into porous substrates follows Washburn’s rule, in general. Here we show that the wicking dynamics are substantially altered when the substrates are structurally anisotropic or soluble in the liquids. First, when the polymer fibers are aligned in one direction, the non-reactive liquid wicks fast along the direction of the fibers following Washburn’s rule while the wicking still occurs across the fibers exhibiting a power law different from Washburn’s rule. Second, when the polymer is soluble in the liquid, the wicking occurs dominantly across the fibers rather than along them, contrary to our intuition. We show that the rate of soluble wicking is determined by viscosity but independent of surface tension of the solution.

3:52PM A38.00005 Wicking-mediated drying in porous media1, MARTA GONALVES, YESEUL KIM, SKKU Advanced Institute of Nanotechnology, Sungkyunkwan University, JIN YOUNG KIM, Research Center for Advanced Materials Technology, Sungkyunkwan University, NAJAF RUBAB, SKKU Advanced Institute of Nanotechnology, Sungkyunkwan University, TAKESHI ASAI, SUNGCHAN HONG, Institute of Health and Sports Science, University of Tsukuba, BYUNG MOOK WEON, School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology, Sungkyunkwan University — When a water droplet is placed on a porous substrate, wicking and drying come up simultaneously. Wicking is a spontaneous liquid flow through porous media by capillarity and drying is a spontaneous vapor flow by vapor diffusion. Despite simultaneity of wicking and drying in porous media, how wicking affects drying is not clear yet. Here we study how wicking dynamics evolves with time and cooperates with drying dynamics by microscopic observations of water droplets on porous materials such as fabrics and papers. We find that wicking at early stages expands the surface area of absorbed water through porous materials, accelerating the evaporation rates. X-ray microscopy is a powerful tool to observe pore networks and water flows inside wicking fabrics. This result is useful to comprehend cooperation and to improve optimization between wicking and drying in porous media.

1Supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (No. 2019R1A1A103301215).

4:05PM A38.00006 Transpiration through hydrogels1, MERLIN ARAGON ETZOLD, M. GRAE WORSTER, PAUL F. LINDEN, DAMTP, University of Cambridge — We present experiments and theory relating to transpiration through hydrogel beads in contact with a water reservoir below and evaporating into air above. Experimentally, we find that saturated hydrogel beads shrink until a steady state is reached in which water flows continuously through the beads. The size of the beads in steady state is very sensitive to the evaporation rate, which depends on the relative humidity and speed of the air, and is measurably sensitive to the pressure in the fluid reservoir. Specifically, the beads are smaller when the evaporation rate is larger or the reservoir pressure lower. Our conceptual 1D model proposes that transport in the hydrogel is driven by gradients in osmotic pressure, therefore by gradients in polymer concentration in the hydrogel, which correspond to gradients in swelling. If the evaporation rate or the bottom pressure changes, the adjustment of this gradient requires the bead to change shape. Smaller beads have larger gradients of osmotic pressure, which drive higher transpiration rates and can draw water against larger hydraulic heads.

1Leverhulme Trust (National Materials Innovation)

4:18PM A38.00007 Carbonized sucrose-coated PDMS sponge for highly efficient and self-cleaning solar evaporator, JAEHYEON LEE, SANG JOON LEE, Pohang Institute of Science and Technology — Solar steam generation is a promising technology for seawater desalination and water purification. However, most state-of-the-art technologies are expensive and suffer from insufficient solar-to-evaporation conversion efficiency and fouling problems, which limit their practical applications. Here, we propose a new superhydrophilic thermally-insulated macroporous membrane (STIMM) composed of carbonized sucrose and polydimethylsiloxane (PDMS) as an efficient solar evaporator. The coupled effect of superhydrophilicity and heat localization of STIMM was found to maximize the solar-to-evaporation conversion efficiency. The highest evaporation rate of 7.702 kg/m^2h, 8.7 times higher than that of only water, was achieved with solar-to-evaporation conversion efficiency of 99.8%. The STIMM was applied to obtain pure drinkable water from saline water with 20 wt% salinity, with a 99.907% desalination efficiency. In addition, the macro-pore size of STIMM enabled self-cleaning with 93.1% of salt rejection rate, which mitigates the fouling problem substantially. The present results demonstrate a highly effective and sustainable solar steam generator, which would be applied for solar thermal desalination and water purification systems.

1NRF-2017R1A2B3005415

Saturday, November 23, 2019 4:40PM - 5:45PM — Session B01 Free-Surface Flows: Interaction with Physical Structures II 2A - Alfonso Ganan-Calvo, Universidad de Sevilla, ETSI

4:40PM B01.00001 Statistics of extreme body motions in nonlinear wave fields, XIANLIANG GONG, YULIN PAN, Department of Naval Architecture and Marine Engineering, University of Michigan — We consider the statistics of extreme body motions in a nonlinear irregular wave field. In addition to the excitation by extreme waves, the extreme motions can be physically resulted from wave-body resonance or parametric excitation. Therefore, their statistics cannot be directly derived from extreme wave statistics which has been studied extensively. The computation of the statistics, however, requires the Monte-Carlo method, which can become computationally intractable (when coupled with high-fidelity simulations) due to the rareness of the extreme events. In this work, we develop a general framework, which enables an efficient resolution of the statistics of extreme body motions in a nonlinear wave field. This leverages a range of physics and learning based approaches, including nonlinear wave simulations, body response simulations, dimension-reduction techniques, sequential sampling and Gaussian process regression (Kriging). The developed method is benchmarked for its effectiveness in accurately resolving the statistics, and applied to study the extreme statistics of ship roll motions in an evolving narrow-banded nonlinear wave field.
4:53PM B01.00002 Experimental study of improving power conversion capability of a floating point absorber with reactive control technique.\textsuperscript{1} YE LI, QIANLONG XU, ZHILIANG LIN, XIAOBO ZHENG, NAOCE. Shanghai Jiao Tong University, BOYIN DING, BENJAMIN CAZZOLATO, ME, University of Adelaide — One of the biggest challenges for wave energy conversion (WEC) system to be competitive in the renewable market is to maintain a high and stable efficiency from stochastic ocean waves with wide spectrum. To solve this issue, there is an emerging need to develop a reactive control system (RCS) for WEC according to incoming wave condition via feedback control. An advanced boundary element method with viscous correction was developed to analyze the system dynamics. Scaled model tests were conducted in the newly developed multiple function towing tank at SJTU to study the power generation of a floating point absorber with RCS. Incoming wave information is well integrated into control loop with a group of gauges. The results show that power output of RCS can be about 2-3 times higher than that of passive control system (PCS). High efficiency spectrum of RC-WEC ranges 1.5 times wider than PC-WEC’s. Viscous effect increases with power generation coefficient, indicating the significance of geometry optimization with control system optimization together. Power output efficiency decreases as wave height increases, indicating that increasing nonlinear hydrodynamics can degrade the performance of linear control.

\textsuperscript{1}This work was supported by National Natural Science Foundation of China (No. 5176115012).

5:06PM B01.00003 Wave attenuation by flexible vegetation. CLINT WONG, University of Oxford, PHILIPPE TRINH, University of Bath, JON CHAPMAN, University of Oxford — The study of fluid flows interacting with vegetative structures in coastal regions presents a significant challenge on account of its multi-scale nature. In this talk, we show how compact asymptotic reductions allow us to study surface waves over submerged vegetative regions. The vegetation is modeled as a collection of elastic cantilever beams in cross-flow and is coupled to fluid momentum equations. Our simplified framework provides some advantages to existing models. For example, for rigid plants over a horizontal bed, previous work on linear waves predicts a quadratic decay in wave amplitude. However, by accounting for plant flexibility, we predict a sub-quadratic decay that agrees more closely with experimental observations. Consideration of a varying depth further demonstrates competition between shoaling and vegetative dissipation.

5:19PM B01.00004 The scaling laws of an exploding liquid cylinder irradiated by an ultrashort X-ray pulse\textsuperscript{1} ALFONSO GANAN-CALVO, Universidad de Sevilla, ETSI, 41092 Sevilla, Spain — A general formulation of the partial destruction of a liquid object in vacuum after the sudden deposition of a very large amount of energy is proposed. That energy instantaneously raises the pressure of a portion of the liquid to extreme values and changes its state, which causes its explosive expansion into vacuum and against the rest of the liquid object. When the deformable object is a liquid capillary cylinder or column, the model reduces to a universal equation for the evolution of the expanding gap between the two sides of the exploding liquid column. The theoretical analysis contemplates two asymptotic stages for small and large times from the initiation of the blast, whose asymptotic solutions are fitted to available experimental data. An universal approximate analytical solution is obtained. A complete dimensional analysis of the problem and an optimal collapse of experimental data reveal that the proposed solution is in remarkable agreement with experiments of a jet exploding after being irradiated by an ultrashort and intense X-ray pulse from a X-ray free electron laser (XFEL).

\textsuperscript{1}Supported by the Ministerio de Economía y Competitividad (Spain), Plan Estatal Retos, project DPI2016-78887-C3-1-R

5:32PM B01.00005 The Dynamics of Liquid Sorbents in Open Capillary Channels\textsuperscript{1} SAMUEL MOHLER, MARK WEISLOGEL, Portland State University, NASA COLLABORATION — Direct contact liquid-gas sorbent beds offer unique benefits for spacecraft air quality control. In a recent ISS technology demonstration experiment (CSELS—Capillary Sorbent), two 16-parallel open capillary channel contactors plumbed in series demonstrated passive ‘thin film’ control, modeling both absorption and desorption functions for a potential low-gravity gas scrubbing system for spacecraft. The open wedge-shaped channels mimic terrestrial falling film reactors by exploiting capillary pressure gradients instead of gravity. In this presentation we highlight the fluid mechanics of the process with and without the effects of CO\textsubscript{2} absorption across the surface. We identify the limits of operation, stability, and transients for systems as functions of wedge geometry and working fluid thermo-physical properties. Rare analytical solutions are found that may be applied to enormous systems of n-parallel channels. The analytical approach serves as the building block for massively parallel systems requiring large surface areas to achieve the desired performance.

\textsuperscript{1}NASA Cooperative Agreement 80NSSC18K0161, CoTR: J. McQuillen, NASA GRC

Saturday, November 23, 2019 4:40PM - 6:11PM –
Session B02 Waves: Surface Waves II

4:40PM B02.00001 Modulation instability and rogue waves for shear flows with a free surface. QING PAN, The University of Hong Kong, ROGER GRIMSHAW, University College London, KWOK WING CHOW, The University of Hong Kong — The evolution of weakly nonlinear, narrow-band wave packets for free surface flows is governed by the nonlinear Schrödinger equation. Rogue waves, unexpectedly large localizations from equilibrium background can occur if the water depth is sufficiently large. In practice, shear currents nearly always occur in oceans, but modeling studies on wave dynamics are usually restricted to the case of linear current. The dynamics of rogue waves in the presence of a linear shear current has been studied in the literature. Generally rogue waves become narrower and with a short period of existence in terms of time if the background plane wave moves against the current. However, the modulation instability can be enhanced when background plane wave moves with the current. Here the investigation is extended to the case of a current with arbitrary vorticity gradient, by enhancing theoretical formulation established earlier by our group. The transient growth rate and the spatial extent of the rogue waves will be reported for two broad classes of velocity profiles, those convex to the right and those convex to the left. And thus knowledge on such focusing mechanisms of free surface waves will be of importance in both nonlinear science and physical oceanography.

4:53PM B02.00002 A new deep water equation for unidirectional surface gravity waves in two dimensions. NAIL S. USSEMBAYEV, KAUST — Using Hamiltonian theory of weakly nonlinear surface waves we derive a set of nonlinear evolution equations describing propagation of unidirectional gravity waves on the surface of a two-dimensional ideal fluid of infinite depth. The proposed equations admit an exact solution in terms of Lambert’s W-function. We compare this solution with the approximate irrotational solution due to Stokes (1847) and the exact rotational solution found by Gerstner (1802). Being irrotational and exact, our solution exhibits a distinctive advantage over the aforementioned classical examples of deep-water gravity waves.
to understand the pair wave creation. Hawking radiation and surface wave-current interactions. Conserved wave activities like pseudomentum and pseudoenergy have been utilized analogous to the celebrated Hawking radiation in Black holes (S.W. Hawking, Nature 1974). Theoretical and numerical investigations on total waves: one oscillating with a positive frequency and the other with a negative frequency. Such a situation is known as the "pair-wave creation", presence of a mean current flowing over a bottom topography have analogies with the Black hole horizon. Refraction of surface waves at the of Dundee, Nethergate, Dundee DD1 4HN, UK, EYAL HEIFETZ, Tel-Aviv University, Tel Aviv 69978, Israel — Surface gravity waves in the for detecting the analogue Hawking effect in a hydraulics laboratory with the aim to support future experimental studies.

The phase velocity pointing in opposite directions. Here we present the correspondence between the Saint-Venant equations for shallow water flows pair had a group velocity and phase velocity both pointing downstream, whereas the second had the unusual property of a group velocity and phase velocity pointing in opposite directions. Here we present the correspondence between the Saint-Venant equations for shallow water flows and furthermore, (ii) the flow speed exceeds the wave speed. At the horizon, the incident shallow water wave splits into two distinct deep water waves: one oscillating with a positive frequency and the other with a negative frequency. Such a situation is known as the "pair-wave creation", analogous to the celebrated Hawking radiation in Black holes (S.W. Hawking, Nature 1974). Theoretical and numerical investigations on total internal reflection and tunneling have been performed using an in-house Higher-order spectral code in order to find deeper analogies between Hawking radiation and surface wave-current interactions. Conserved wave activities like pseudomentum and pseudoenergy have been utilized to understand the pair wave creation.

The effect of wind on wave shape: Shallow water, THOMAS ZDYRSKI, FALK FEDDERSEN, Univ of California - San Diego — Wave shape (e.g., wave skewness and asymmetry) impacts sediment transport, beach morphology, and ship safety. Previous work by the authors showed that wind (via changes in surface pressure) affects wave shape in intermediate and deep water. This effect was most pronounced as the depth (kh) decreased. Here, this work investigates the interaction of wind and wave shape in shallow water. A multiple-scales analysis is applied to waves propagating over a shallow (kh ≪ 1), flat bottom with a variety of wind-induced surface-pressure profiles, such as Jeffreys-type and generalized Miles-type. The shallow depth enhances the influence of wind on wave shape and intensifies the waves’ second-harmonic modes. The results are compared to previous wave-tank experimental data and numerical simulation results.

Wave drag on asymmetric bodies, GRAHAM BENHAM, JEAN-PHILIPPE BOUCHER, ROMAIN LABB, MICHAEL BENZAQUEN, CHRISTOPHE CLANET, LaDHyX, Ecole Polytechnique — More than a century ago, Michell derived an integral formula for the wave resistance on a body, using the approximation of a slender body in an irrotational, inviscid fluid (Michell 1898). The major shortcoming of this formula is that, due to the reversibility of the steady potential flow formulation, it does not distinguish the difference in wave drag when an object with front-back asymmetry moves forwards or backwards. However, it is well known that an asymmetric body with a sharp leading edge and a rounded trailing edge produces a smaller wave disturbance moving forwards than backwards, and this is reflected in the wave drag coefficient. In this talk, we discuss recent experimental observations investigating the effects of body asymmetry on wave drag, and show that these effects can be replicated by modifying Michell’s theory to include the growth of a symmetry-breaking boundary layer. We demonstrate that asymmetry can have either a positive or a negative effect on drag, depending on the depth of motion and the Froude number. We discuss the implications and scope of this work in the context of sports physics, including the design of rowing, kayak and canoe boats.

Jetting in large amplitude axisymmetric capillary gravity waves, RATUL DASGUPTA, SAWSATA BASAK, PALAS KUMAR FARSOIYA, Indian Institute of Technology Bombay — The phenomenon of jetting and accompanying droplet ejection is known to occur in many fluid dynamical situations involving collapse of a gaseous cavity at a liquid-gas interface. In a recent study of free, capillary-gravity oscillations on a quiescent cylindrical pool of liquid [Farsoiya et al J. Fluid Mech., 857, pp. 80-110 (2017)], it has been shown using DNS that jetting may be obtained with an initial interfacial perturbation in the form of a single Bessel mode. Sufficiently large values of wave steepness lead to the formation of a jet at the axis of symmetry that can eject droplets from its tip. In this study, we formulate the solution to the weakly nonlinear, inviscid-irrotational, initial-value problem in axisymmetric cylindrical coordinates using wave steepness as a small parameter and obtain a third order solution for the free surface profile. It is seen that the weakly nonlinear solution is able to describe the onset of jet formation. A detailed discussion of capillary effects along with comparisons with Direct Numerical Simulations, will be presented.

Jetting in large amplitude axisymmetric capillary gravity waves, RATUL DASGUPTA, SAWSATA BASAK, PALAS KUMAR FARSOIYA, Indian Institute of Technology Bombay — The phenomenon of jetting and accompanying droplet ejection is known to occur in many fluid dynamical situations involving collapse of a gaseous cavity at a liquid-gas interface. In a recent study of free, capillary-gravity oscillations on a quiescent cylindrical pool of liquid [Farsoiya et al J. Fluid Mech., 857, pp. 80-110 (2017)], it has been shown using DNS that jetting may be obtained with an initial interfacial perturbation in the form of a single Bessel mode. Sufficiently large values of wave steepness lead to the formation of a jet at the axis of symmetry that can eject droplets from its tip. In this study, we formulate the solution to the weakly nonlinear, inviscid-irrotational, initial-value problem in axisymmetric cylindrical coordinates using wave steepness as a small parameter and obtain a third order solution for the free surface profile. It is seen that the weakly nonlinear solution is able to describe the onset of jet formation. A detailed discussion of capillary effects along with comparisons with Direct Numerical Simulations, will be presented.

Optimization of a highway infrastructure design through a combined computational fluid dynamics and evolutionary algorithm framework, PENG ZHANG, ANH-VU VO, DEBRA LAEFER, Tandon School of Engineering, New York University, MAURIZIO PORFIRI, Department of Mechanical and Aerospace Engineering and Department of Biomedical Engineering, Tandon School of Engineering, New York University — Design of highway infrastructure significantly influences the performance and fuel consumption of vehicles. However, standard design practice heavily relies on empirical knowledge, based on difficult-to-generalize and cost-prohibitive experiments. Here, we propose a global optimization framework based on computational fluid dynamics (CFD) and evolutionary algorithms to identify the optimal design of a highway infrastructure that minimizes aerodynamic drag on moving vehicles. Although the framework is broadly-applicable, we investigate the design using an actual section of highway with vertical side walls. The highway’s geometry was captured by terrestrial laser scanning and is used to generate the computational domain for three-dimensional CFD. The drag on the vehicles is estimated through CFD using a k-ε turbulence model. Reduction in the aerodynamic drag is attained by enhancing the interaction between the vehicles and the vertical side walls through the addition of solid slabs to the walls. The optimization process is performed through an evolutionary algorithm, which iteratively evolves the size and positioning of the added slabs toward an optimal design. Our results demonstrate that the proposed computational framework can inform future highway infrastructure designs.

This research was supported by the Brookwood Foundation.
4:53PM B03.00002 Unsteady Aerodynamics of Roll-Tacking and Roll-Gybing in an Olympic Class Sailboat, SARAH MORRIS, C. H. K. WILLIAMSON, Cornell University — In this research, we study unsteady sail motion techniques used by Olympic sailors to increase sailboat propulsion. One such technique is for sailors to use bodyweight movements to roll the boat about its longitudinal axis. This motion is used especially for turning the boat in light winds, by either “roll-tacking” (upwind sailing) or “roll-gybing” (downwind sailing). When roll-tacking and roll-gybing, sailors dynamically roll the boat to increase their speed and propel their boats faster than using wind alone; this is in contrast to flat-tacking and flat-gybing, wherein the sailor keeps the boat level (and mast vertical) while turning. Each of these motions is characterized in on-the-water experiments using a Laser sailboat equipped with a GPS, IMU, wind sensor and GoPro camera array. Bringing these motions into the laboratory, we study the underlying vortex dynamics that are responsible for generating extra propulsion in roll-tacking and roll-gybing.

5:06PM B03.00003 Effect of Hybrid-Heave Motions on the Lift of Oscillating Airfoils1, JAY YOUNG, SARAH MORRIS, C. H. K. WILLIAMSON, Cornell University — Inspiration for a number of unsteady airfoil dynamics studies has been drawn from animal locomotion. In the present research, we instead employ a sports-mimetic approach, studying unsteady airfoil motions inspired by sail dynamics. One such technique is “sail flicking,” whereby sailors use their bodyweight motion to roll the boat about its longitudinal axis, flicking the sail periodically. Because sailors do not sail directly into the wind, the boat travels at an angle relative to the apparent wind it experiences. A “flicking” sail will therefore oscillate at non-perpendicular angles to the incoming flow, in a motion we call “hybrid-heave.” We study these hybrid-heave motions with a NACA 0012 airfoil. We find both a “high-lift” and “low-lift” mode. The high-lift mode delivers significant lift increase compared to that of a static airfoil. We study the case of the two distinct modes in terms of instantaneous angle of attack, speed, and vortex dynamics.

5:19PM B03.00004 Effect of Spherical Depressions on Hatchback Cars: A CFD Study, SOURAJIT BHATTACHARJEE, VISHESH KASHYAP, PRIYANSHU MITTAL, B.B. ARORA, Delhi Technological University — A major reason for the development of form drag is vortices generated due to flow separation. This greatly affects the aerodynamics of light vehicles such as hatchback cars. Flow separation may be delayed by the creation of spherical depressions at various locations on the vehicle body, which could lead to a decrease in the form drag. Through a series of studies, the effect of spherical depressions of aspect ratios 2, 4, 6 and 8 was studied separately on the bonnet, doors and roof of a generic hatchback car. 3-D CFD analyses were performed using ANSYS Fluent using a validated computational model. The trends for drag force and drag coefficient with aspect ratio were observed to vary with change in the location of the depressions. While an aspect ratio of 4 delivered the greatest decrease in drag for the bonnet, an increasing trend was observed for the other 2 locations. At higher aspect ratios, the drag was observed to be higher than for the car without depressions. The studies demonstrated the effect of spherical depressions on hatchback cars and elaborated on the importance of location as well as aspect ratio as factors to be considered for low aerodynamic drag.

5:26PM B03.00005 Aerodynamic Assessment of Cross-Wind Airwake Characteristics at Ship Helodeck: A Simplified Approach, SHRISH SHUKLA1, S. N. SINGH2, S. S. SINHA3, Indian Institute of Technology Delhi, R. VIJAYAKUMAR, Indian Institute of Technology Madras — The combined ship-helicopter operation nearby small naval vessels has historically been one of the most challenging tasks. The complexity of this task primarily depends on the ship superstructure, helicopter aerodynamics, and cross-wind conditions. An early assessment of the overall airwake characteristics is one of the challenging tasks. The First-Of-Class Flying Trials are one of the most common methods to assess the resultant airwake characteristics over helodeck. These trials are conducted post-construction of the ship and mainly based on qualitative ratings of the pilot. Also, a wide range of wind conditions cannot be covered, and the scope of further design modification is limited. Thus, there is a strong need to assess the impact of cross-wind conditions on the ship helodeck at the initial design stages. We present an early-stage simplified approach to estimate the aerodynamic impact of cross-winds over ship helodeck with reasonable accuracy. A Reynolds-averaged-Navier-Stokes based parametric analysis has been conducted for six cross-wind conditions to understand the cross-wind phenomena over a simplified frigate ship helodeck. The paper reports the cross-wind airwake assessment based on the criteria set developed by mean velocity gradients and turbulence intensity.

5:45PM B03.00006 Busemann-Sears-Haack hybrid geometries applied toward supersonic vehicles for improved wave drag performance, ANDREW SKLAR, ZVI RUSAK, Rensselaer Polytechnic Institute — We present a new configuration for supersonic aircraft fuselages. It is first demonstrated that the commercial Fluent code provides mesh converged, valid results for inviscid supersonic flows around various configurations compared to classical predictions. Then, by adapting a hybrid geometry of the Busemann bpline shape to a span-wise split Sears-Haack body in the region between the bodies, we present a physically feasible two-body configuration that reduces shock wave interference between the two bodies and lowers the wave drag per volume of a given fuselage volume and length. The reduction is about 50% when compared to the Sears-Haack body with same volume and length. In addition, when applied to non-enclosed geometries, the Busemann bpline experiences none of the wave drag spikes and hysteresis that were found with bpline configurations. Preliminary studies into effects of viscosity for supersonic flows at high Reynolds numbers show that the total drag is limited to 25% greater than inviscid flow results. This effect has also been extended to a triple body configuration, which further cuts the drag per volume to less than 40% of the equivalent Sears-Haack body.

5:58PM B03.00007 String Stability in Energy-Saving Formation Flight1, PHILIPPE CHATELAIN, JAMES RIEHL, ESTEBAN HUFSTEDLER, JULIEN HENDRICKX, Université Catholique de Louvain — Formation flight of fixed-wing aircraft provides the opportunity to save substantial energy by exploiting the upwash created by vortical wakes. For the most efficient flight, the aircraft need to carefully track the relative wake position of their leader, which may move due to turbulence or maneuvering of the leader. The tracking errors may grow as the disturbances propagate to aircraft further downstream, resulting in ‘string instability’. This is more commonly examined in automobile platoons, and has not been adequately examined with respect to efficient aircraft formations. We discuss some pitfalls and trade-offs involved in designing string stable controllers for this system, and describe a controller that achieves both string stability and energy efficiency.

1This project has received funding from the European Research Council under the European Unions Horizon 2020 research and innovation program, grant agreement No 725627.
compared with the experiments. The secondary flow effect in the transport equation. The numerical results of depth-averaged particle concentration profile are presented and the effect of the particle accumulation, we extract the particle flux as a function of local particle concentration from the experiments and include system mathematically by considering the depth-averaged particle transport equation and suspension balance model. To capture the physical lead to the gradual accumulation of particles on the advancing oil-air interface. This particle accumulation results in the fingering of an otherwise stable fluid-fluid interface. While the previous works by Xu and colleagues have focused on the resultant instability, one unexplored lead to inertia, is key to triggering hysteresis. To understand this link between friction and hysteresis, we measured the suspension rheology close even without inertia. By using microscale particles whose interparticle friction can be turned off, we show that microscopic friction, conversely to the flow onset for frictional and frictionless suspensions. We show that the flow rule for frictionless particles is monotonous with a power law of exponent 0.05 ± 0.05, in close agreement with the previous theoretical prediction, 0.35. By contrast, the flow rule for frictional particles suggests a velocity-weakening behavior. These findings show that hysteresis can occur in particulate media without inertia, and by highlighting the role of microscopic friction, it questions the intimate nature of this phenomenon.

Fernando Vereda acknowledges funding from the University of Granada through the Brown/CASA-UGR Research Collaboration Fund and by MINECO MAT 2016-78778-R project (Spain)

4:53PM B04.00002 Self-similarity in particle accumulation at fluid-fluid interface. LI WANG, YUN CHEN, RUI LUO, SUNGYON LEE, University of Minnesota — When the mixture of viscous oil and non-colloidal particles displace air between two parallel plates, the coupled effects of shear-induced migration of particles and the secondary flow near the interface lead to the gradual accumulation of particles on the air-oil interface. This particle accumulation results in the fingering of an otherwise stable fluid-fluid interface. While the previous works by Xu and colleagues have focused on the resultant instability, one unexplored yet striking feature of the experiments is the self-similarity in the concentration profile of the accumulating particles. In this talk, we model the system mathematically by considering the depth-averaged particle transport equation and suspension balance model. To capture the physical effect of the particle accumulation, we extract the particle flux as a function of local particle concentration from the experiments and include the secondary flow effect in the transport equation. The numerical results of depth-averaged particle concentration profile are presented and compared with the experiments.

5:06PM B04.00003 Surface instability of shear-thickening suspensions down an inclined plane. BAPTISTE DARBOIS TEXIER, BLOEN METZGER, HENRI LHUISSIER, YOEL FORTERRE, Aix-Marseille University, IUSTI-CNRS UMR 7343, 13453 MARSEILLE, FRANCE TEAM — Shear-thickening of dense suspensions is one of the most appealing phenomena of science festivals. The dramatic increase of the viscosity with the shear stress is now understood as a frictional transition occurring above a critical stress set by a repulsive interaction between particles. Here, we investigate the stability of a thin film of cornstarch suspension flowing down an inclined plane. At low packing fractions, the film is unstable above a critical Reynolds number given by the well-known Kapitza criterion for Newtonian fluids. However, at high packing fractions, as shear-thickening becomes discontinuous, a new instability emerges at Reynolds numbers much smaller than the Kapitza threshold. We show that this instability arises from the characteristic S-shape of the rheological law.

5:19PM B04.00004 Particle-induced miscible fingering. RUI LUO, YUN CHEN, SUNGYON LEE, University of Minnesota Twin Cities — We experimentally inject silicone oil into the mixture of oil and non-colloidal particles inside a Hele-Shaw cell, to investigate the connection between miscible fingering and the flow structure that develops in the thin gap. Previous studies with pure fluids have demonstrated that the onset of miscible fingering coincides with the transition from a smooth tongue-like structure to a sharp front between invading and defending fluids inside the thin gap. Our current experiments reveal the same general behavior at the onset of miscible fingering, which we capture qualitatively using a continuum model. However, beyond the onset, we observe distinctly different morphologies of miscible fingering, which depend on the ratio of the gap thickness to particle diameter. We present the new quantitative measurements that highlight these differences and discuss how the wall confinement may alter the particle dynamics and the resultant fingering patterns.

5:32PM B04.00005 Experimental and numerical studies of particle-laden fluid flows over a porous media model. EILEEN HAFFNER, CHANGWOO KANG, University of Illinois at Chicago, NINA SHAPLEY, Rutgers University, The State University of New Jersey, Laboratory, Seattle, University of Illinois at Chicago — Suspension flows have been extensively studied through various experimental techniques and numerical simulations but only in smooth channels. On the other hand, flow of pure Newtonian fluid over porous media has been the topic of several investigations. However, to the best of the author’s knowledge, how these two engineering systems relate to one another is still unknown. This study was conducted to examine the interaction between various suspensions over a porous media model. Two experimental techniques were utilized, particle image velocimetry (PIV) and nuclear magnetic resonance (NMR) imaging. The PIV data provided two dimensional velocity vector fields which was used to extract slip velocity and shear rate at the interface between the free flow region and the porous media for dilute suspension flows. The NMR measurements provided three-dimensional velocity and concentration information through and above the porous media for higher concentrated suspensions. It was found that the slip velocity, shear rate, and concentration profiles are strongly dependent on the suspension concentrations as well as the geometry and properties of the porous media. Theoretical simulations were developed and compared to the experimental results, showing good agreement in the free flow region.}

National Science Foundation award #1854376 and Army Research Office award #W911NF-18-1-0356

5:45PM B04.00006 Hysteresis in viscous suspensions. HUGO PERRIN, CECILE CLAVAUD, BLOEN METZGER, CNRS - IUSTI, MATTIEU VYAYART, EPEF, YOEL FORTERRE, CNRS - IUSTI — Hysteresis is a major feature of the solid-liquid transition in granular materials. This property, by allowing metastable states, can potentially yield catastrophic phenomena such as landslides. The origin of hysteresis in granular flows is still debated. However, most mechanisms put forward so far rely on inertia at the particle level. Here, we study the avalanche dynamics of non-Brownian suspensions in slowly rotating drums and reveal large hysteresis of the avalanche angle even without inertia. By using microscale particles whose interparticle friction can be turned off, we show that microscopic friction, conversely to inertia, is key to triggering hysteresis. To understand this link between friction and hysteresis, we measured the suspension rheology close to the flow onset for frictional and frictionless suspensions. We show that the flow rule for frictionless particles is monotonous with a power law of exponent 0.37 ± 0.05, in close agreement with the previous theoretical prediction, 0.35. By contrast, the flow rule for frictional particles suggests a velocity-weakening behavior. These findings show that hysteresis can occur in particulate media without inertia, and by highlighting the role of microscopic friction, it questions the intimate nature of this phenomenon.
**5:58PM B04.00007 Mixing in sheared suspensions**, REGIS TURUBAN, HENRI LHUISSIER, BLOEN METZGER, JUSTI CNRS UMR 7343 Marseille France, GEP TEAM — Mixing occurs spontaneously in sheared suspensions even at low Reynolds number. The presence of particles induces disorder which lead to exponential elongations within the fluid: concentration levels thus quickly spread and decay. We experimentally characterize the evolution of the concentration PDFs of a blob of fluorescent dye initially injected in an index-matched suspension. High precision optical imaging technics reveal for the first time the finest spatial details of the concentration field possibly generated by this chaotic flow (Batchelor scales). We find that at short times, the evolution of the concentration PDFs are correctly predicted by a model based solely on the stretching kinematics. At longer times, we show that to predict the experimental observations, the model should also include the effect of coalescence between adjacent lamellae of dye.

**Saturday, November 23, 2019 4:40PM - 6:11PM — Session B05 Compressible Flows: Shock Waves and Explosions**

**4:40PM B05.00001 Scale-invariant Homentropic Compressible Flows and Their Application to the Noh Problem**, JESSE GIRON, Arizona State University, SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — The purpose of this work is to examine the group invariance properties of the inviscid Euler equations, subject to an equation of state (EOS) where the fluid pressure is regarded solely as an arbitrary function of the fluid density. We derive the conditions under which the resulting homentropic Euler equations and associated shock jump conditions are invariant under scaling groups. The invariance properties of these relations are used to construct a Noh-like solution featuring a constant velocity stagnation shock. For this solution to exist, we demonstrate that the EOS must satisfy a transcendental algebraic relation.

1This work was supported by the U.S. Department of Energy (DOE) through the Los Alamos National Laboratory. Los Alamos National Laboratory is operated by Triad National Security, LLC, for the National Nuclear Security Administration of the DOE (contract number 89233218CNA000001).

**4:53PM B05.00002 Quasi-Similar Converging Shock Flows for a Mie-Gruneisen Equation of State**, EMMA SCHMIDT, SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — The Guderley hydrodynamic test problem is an example of a self-similar, scale-invariant solution of the inviscid Euler equations. It consists of an infinitely strong shock wave converging to an axis or point of symmetry, where the defined flow has the notable property of being independent of a system of units. A key piece of the self-similar analysis is selecting a material that exhibits scale invariant behavior. The symmetry properties of the Euler equations have been used by several authors to determine the general form of an equation of state (EOS) that permits the existence of scale-invariant solutions. However many widely used EOSs, such as the Mie-Gruneisen EOS for crystalline solids, are not of the required form. We propose a joint similarity-perturbative analysis that modifies the classical self-similar solutions of the Euler equations to represent non-ideal material behavior, and perform the quasi-similar analysis of the Guderley problem in a non-ideal material defined by the Mie-Gruneisen EOS.

**5:06PM B05.00003 Numerical modeling of solid-cluster evolution applied to the nanosecond solidification of water through ramp and shockwave compression**, DANE STERBENTZ, University of California, Davis, PHILIP MYINT, Lawrence Livermore National Laboratory, JEAN-PIERRE DELPLANQUE, University of California, Davis, JONATHAN BELOF, Lawrence Livermore National Laboratory — Classical nucleation theory (CNT) is a promising way to predictively model the sub-microsecond kinetics of rapid phase transitions that occur under ramp or shock compression, such as the suite of experiments performed over the past two decades on the solidification of liquid water to the high-pressure ice VII phase. We model the liquid water–ice VII phase transformation in these hydrodynamic-loading experiments using a numerical discretisation scheme to solve the Zel’dovich–Frenkel partial differential equation (a fundamental CNT-based kinetic equation that describes the statistical time-dependent behavior of solid cluster formation and accounts for transience in the nucleation kinetics) as well as through hydrodynamics simulations. We have also developed a new dimensionless parameter that may be applied a priori to predict whether or not transient nucleation will be important in a given ramp- or shock-compression experiment.

1This work was performed under the auspices of the U.S. Department of Energy (DOE) by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344. We thank A. Arsenlis, D.P. McNabb, B. Wallin, and C. Clouse for funding and project support. Program support was also provided by the DOE NNSA Laboratory Residency Graduate Fellowship.

**5:19PM B05.00004 Shock-wave structure according to the Navier–Stokes–Fourier constitutive relations**, FRANCISCO J. URBIE, ROSA M. VELASCO, Universidad Autonoma Metropolitana — The Navier–Stokes–Fourier constitutive equations are used to study plane shock–waves in dilute gases. We use the soft sphere model in which the viscosity and thermal conductivity are proportional to a power of the local temperature: \( \eta, \kappa \sim T^\sigma \), \( \sigma \) being the viscosity index. We show that the experimental normalized density profiles for argon can fit with the viscosity index. Results form the direct simulation Monte Carlo method and with transport coefficients obtained from ab initio potentials are also considered.

**5:32PM B05.00005 Non-classical behavior in shock-compressed gas mixtures**, PATRICK WAYNE, PETER VOROBIEFF, CAROLINA SHAHEEN, DANIEL FREELONG, C. RANDALL TRUMAN, University of New Mexico — Rankine-Hugoniot (R-H) equations, while assuming a calorically perfect gas, provide a good approximation for real gases undergoing shock compression. Our experiments focus on problems that arise when an attempt to predict post-shock gas properties is made for a non-reacting mixture of two gases with highly disparate properties (hydrogen and sulfur hexafluoride). Although the range of Mach numbers we consider is quite modest (1.2 to 2.0), we observe major deviations from the behavior that classical models predict. The latter include R-H equations in combination with either Dalton’s or Amagat’s law to characterize the gas mixture. Moreover, we observe these deviations to persist on time scales much longer than those historically associated with non-equilibrium behavior due to shock front passage.

1We acknowledge prior support from the National Nuclear Security Administration (NNSA) grant DE-NA-0002913.
Maxwell Koger, Conor Pace, Michael Vu, Oleg Goushcha, Manhattan College — A unique propulsion system has been assembled in which a flow is driven using a fan actuated at its outer diameter rim. This fan configuration eliminates the need for a shaft and a centerline hub required by the conventional fan designs allowing for an undisturbed flow through the centerline region. Also, in our design the fan blades extend from the rim inward towards the centerline. Blades at the outer radius are connected to the rim. The inner radius blade tips are located in a relatively low-flow region associated with small tangential velocity of the fan near the centerline. Therefore, the tip vortices and the associated noise produced by the tip vortices should be minimized compared to the conventional propeller configuration where blade tips are located in a high-tangential velocity region associated with the outer radius. We present an experimental study of noise signature from the tandem airfoil configuration. Near-field measurements have been carried out to understand the mechanism leading to the noise reduction by studying how surface pressure fluctuations vary over the serration root and tip planes. The near-field hydrodynamic analysis obtained using turbulence quantities are estimated from the refractive images and show variations as a function of confinement thickness.

Funding provided by DTRA grant HDTRA1-18-1-002 with PI: Dr. Peter Vorobieff at UNM and program manager Dr. Paul Tandy and funding from the US Air Force via IS4S.

Funding provided by EPSRC and University of Bristol under the EPSRC Doctoral Prize Scheme is gratefully acknowledged.
5:19PM B06.00004 The Propeller-Induced Cavitation Noise Source: Experimental Measurements and Numerical Solutions. DUNCAN MCINTYRE, MOSTAFA RAHIMPOUR, University of Victoria, GIORGIO TANI, FABIANA MIGLIANTI, MICHELE VIVIANI, University of Genoa, ZUOMIN DONG, PETER OSHKAI, University of Victoria — Propeller-induced cavitation dominates the mid-to high-frequency range of underwater radiated noise emitted by ships, representing a significant threat to marine ecosystems. The development of mitigation strategies for noise pollution requires predictive models, which are challenging to develop due to the varied, multiscale, and multi-physical nature of the phenomenon. One promising technique for predicting the detailed behaviour of the propeller cavitation noise relies on the use of URANS solutions of the cavitating flow with a volume-of-fluid cavitation model as an input for acoustic modelling with a porous surface formulation of the Ffowcs Williams-Hawking analogy. We present an application of this methodology involving the reproduction of model-scale experiments in a cavitation tunnel. We focused on ten loading conditions of a controllable pitch propeller that resulted in four distinct regimes of cavitation. Performance of the hydrodynamic model varied depending on the cavitation regime. Cavities attached to propeller blades well, but regimes involving cavities within shed vortices were not reproduced well. The numerical model was effective in predicting the shapes of acoustic spectra, but the absolute sound levels were overpredicted.

We acknowledge the support of the Clean Transportation Initiative of Transport Canada, the Natural Sciences and Engineering Research Council of Canada, Compute Canada, and our industrial collaborators.

5:32PM B06.00005 Noise Reduction Mechanism of a High-Lift Airfoil’s Leading-Edge Device. RINIE AKKERMANS, PAUL BERNICKE, TU Braunschweig — Airframe noise is a major part of the total noise produced by an aircraft during its landing phase, of which the wing’s leading-edge device (i.e., slot) is a major contributor. In this contribution, the noise reduction of such slat devices is investigated by means of Overset-LES simulations. It solves the compressible Navier-Stokes equations in perturbation form over a background flow, supplemented by a sub-filter-stress model. Two geometries are considered, i.e., a reference and a modified long-cord slat geometry. The effect of the long-cord slat on the turbulent sound sources is investigated by mainly considering turbulence statistics and span-wise coherence length in the slat cove region. Furthermore, acoustic far-field propagation reveals the influence of long-cord slat on the directivity. Results show that the noise reduction is mainly resulting from a shielding effect by the long-cord slat, rather than modification of the turbulent sound source in the slat cove region itself.

5:45PM B06.00006 An experimental analysis on efficacy of perforated plates of varying orifice quantity. T DHANACHANDRAN THANAPAL, JINLIANG HENG, BASMAN ELHADIDI, WAI LEE CHAN, Nanyang Technological University — Combustion engines typically generate significant noise signatures, and the latter constitutes a source of noise pollution. One method of reducing such noise is by disrupting the acoustic waves using acoustic dampers, which in this work is in the form of perforated plates. As air flows through the orifices of the plates, an unsteady jet flow is generated, which will further interact with the acoustic noise and, through viscous dissipations, convert the acoustic fluctuations into non-radiating vortical fluctuations. With the recognition of this fundamental mechanism, the objective of this work is to understand the effectiveness of the perforated plates as acoustic dampers. Fast Fourier transformation was used to analyse the resulting microphone data, showing that the single-orifice plate functioned as an acoustic resonator instead of a damper. In contrast, the multiple-orifice plate functioned as acoustic damper, producing approximately 50% reduction in noise. This finding is consistent with prior investigation where plates with higher open area ratios perform better at reducing acoustic noise. Both single- and multiple-orifice plates were relatively insensitive to the bias flow rate, although the single-orifice plate performance did improve slightly when the bias flow rate was increased.

5:58PM B06.00007 Numerical analysis of the characteristics of Helmholtz resonators with multiple necks. JINLIANG HENG, T DHANACHANDRAN THANAPAL, WAI LEE CHAN, BASMAN ELHADIDI, Nanyang Technological University — Helmholtz resonators are commonly used in combustion engines to reduce engine noise and tackle combustion instability issues. A resonator, typically consisting of a fixed volume cavity and a neck has a resonant frequency that is a function of the speed of sound, neck cross-sectional area, cavity volume, and neck length. For a resonator with multiple necks, the analytical expression is able to determine a single value for its resonant frequency. However, in this work involving a cavity with multiple necks, experiments have shown that multiple frequencies are present. Subsequent numerical simulations utilizing the Lattice Boltzmann method also yield reasonable agreement with experiments, suggesting that the multiple-frequency phenomenon may be attributed to interactions between different necks, and may be specific to the current experimental setup. Further simulations with sinusoidal and pulse acoustic source will be performed for further insights to the flow associated with a resonator with multiple necks, from which the multiple-frequency phenomenon may be explained.

Saturday, November 23, 2019 4:40PM - 5:58PM – Session B07 Suspensions: General IV 211 - Bryan Quaife, Florida State University

Awards for Oscillatory Flow

practical ability for oscillatory flow, compared to steady flow, for the analysis of smaller sample volumes. I will also discuss the impact of the channel depth, width, pressure gradient and frequency of oscillations on the alignment in the system. For the specific application of pathogen detection via binding of M13 bacteriophage, I will explore with the narrow channel flow equations, the latter modified to incorporate mechanical anisotropy induced by the particles. The linear dichroism orientation dynamics of a suspension of slender microscopic fibres. The model couples the Fokker-Planck equation for Brownian suspensions and, more generally, extend to the rheology of mixed suspensions having particles with different rigidities, as well as offering possibilities for volume fraction of rigidified cells. These results are relevant to better characterize SCA, provide useful insights relevant to blood transfusions, and, more generally, extend to the rheology of mixed suspensions having particles with different rigidities, as well as offering possibilities for developments in the field of soft material composites.

ABHILASH REDDY MALIPEDDI, KAILIK SARKAR, The George Washington University — We compute the shear-induced gradient-diffusivity of red blood cells (RBC) from direct numerical simulation using a dynamic structure factor approach. Macro-scale phenomena, such as diffusivities, result from the micro-scale dynamics of RBCs. RBCs in shear flow exhibit extremely rich and complex motion such as tank-treading, tumbling and swinging. They affect the shear-induced diffusivity analyzed here. As the shear rate increases, the diffusivity increases initially due to the increased deformability of the cells, and thereby enhanced interactions between them. On further increase of the shear rate, a transition point is reached where the diffusivity briefly decreases before increasing again. The decrease corresponds to the transition of the single cell dynamics from the tumbling regime to the tank-treading regime. During tumbling of an RBC, due to the larger swept volume of the tumbling shape, its effective size and correspondingly the length scale for diffusion is larger than what is indicated by the cell volume. In the tank-treading regime, this effect is absent in lowered diffusivity.

ABHILASH REDDY MALIPEDDI, KAILIK SARKAR, The George Washington University — We compute the shear-induced gradient-diffusivity of red blood cells (RBC) from direct numerical simulation using a dynamic structure factor approach. Macro-scale phenomena, such as diffusivities, result from the micro-scale dynamics of RBCs. RBCs in shear flow exhibit extremely rich and complex motion such as tank-treading, tumbling and swinging. They affect the shear-induced diffusivity analyzed here. As the shear rate increases, the diffusivity increases initially due to the increased deformability of the cells, and thereby enhanced interactions between them. On further increase of the shear rate, a transition point is reached where the diffusivity briefly decreases before increasing again. The decrease corresponds to the transition of the single cell dynamics from the tumbling regime to the tank-treading regime. During tumbling of an RBC, due to the larger swept volume of the tumbling shape, its effective size and correspondingly the length scale for diffusion is larger than what is indicated by the cell volume. In the tank-treading regime, this effect is absent in lowered diffusivity.

YOUNG, New Jersey Institute of Technology, DAVID WOOD, University of Minnesota, JOHN HIGGINS, Massachusetts General Hospital, ZHANGLI FENG, University of Notre Dame, HOWARD STONE, Princeton University — Sickle cell anemia (SCA) is a disease that affects red blood cells (RBCs) within blood. Healthy RBCs are highly deformable objects that under flow can penetrate blood capillaries smaller than their typical size. In SCA there is an impaired deformability of some cells, which are much stiffer and with a different shape than healthy cells, and thereby affect regular blood flow. It is known that blood from patients with SCA has a higher viscosity than normal blood. However, it is unclear how the rigidity of cells is related to the viscosity of blood, in part because SCA patients are often treated with transfusions of variable amounts of normal RBCs and only a fraction of cells will be stiff. Hence, we report systematic viscosity measurements and numerical simulations of a suspension varying the fraction of RBCs affected by increased rigidity within a suspension of healthy cells. Our results show that there is a rheological signature within blood viscosity to clearly identify the fraction of rigidified cells within healthy deformable cells down to a 5% volume fraction of rigidified cells. These results are relevant to better characterize SCA, provide useful insights relevant to blood transfusions, and, more generally, extend to the rheology of mixed suspensions having particles with different rigidities, as well as offering possibilities for developments in the field of soft material composites.

GEMMA CUPPLES, DAVID SMITH, University of Birming-
ham, MATTHEW HICKS, Linear Diagnostics Ltd, ROSEMARY DYSON, University of Birmingham — Flow linear dichroism is a biophysical spectroscopic technique that exploits the shear-induced alignment of elongated particles in suspension. This talk is focussed around the broad aim of optimising the sensitivity of this technique by improving the alignment of these particles, with a specific application of a handheld synthetic biotechnology prototype for waterborne-pathogen detection. I will describe a model of steady and oscillating pressure-driven channel flow and orientation dynamics of a suspension of slender microscopic fibres. The model couples the Fokker-Planck equation for Brownian suspensions with the narrow channel flow equations, the latter modified to incorporate mechanical anisotropy induced by the particles. The linear dichroism signal is estimated through integrating the perpendicular components of the distribution function via an appropriate formula that takes the bi-axial nature of the orientation into account. For the specific application of pathogen detection via binding of M13 bacteriophage, I will explore the impact of the channel depth, width, pressure gradient and frequency of oscillations on the alignment in the system. I will also discuss the practical ability for oscillatory flow, compared to steady flow, for the analysis of smaller sample volumes.

This work was supported by a BBSRC Industrial CASE Studentship (BB/L015587/1) and an EPSRC Healthcare Technologies Challenge Award (Rapid Sperm Capture EP/N021096/1).

YOUNG, New Jersey Institute of Technology, DAVID WOOD, University of Minnesota, JOHN HIGGINS, Massachusetts General Hospital, ZHANGLI FENG, University of Notre Dame, HOWARD STONE, Princeton University — Sickle cell anemia (SCA) is a disease that affects red blood cells (RBCs) within blood. Healthy RBCs are highly deformable objects that under flow can penetrate blood capillaries smaller than their typical size. In SCA there is an impaired deformability of some cells, which are much stiffer and with a different shape than healthy cells, and thereby affect regular blood flow. It is known that blood from patients with SCA has a higher viscosity than normal blood. However, it is unclear how the rigidity of cells is related to the viscosity of blood, in part because SCA patients are often treated with transfusions of variable amounts of normal RBCs and only a fraction of cells will be stiff. Hence, we report systematic viscosity measurements and numerical simulations of a suspension varying the fraction of RBCs affected by increased rigidity within a suspension of healthy cells. Our results show that there is a rheological signature within blood viscosity to clearly identify the fraction of rigidified cells within healthy deformable cells down to a 5% volume fraction of rigidified cells. These results are relevant to better characterize SCA, provide useful insights relevant to blood transfusions, and, more generally, extend to the rheology of mixed suspensions having particles with different rigidities, as well as offering possibilities for developments in the field of soft material composites.

1This project is partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436

1This work was partially supported by NSF Grant CBET-1706436
4:40PM B08.00001 Flying in a superfluid: starting flow past an airfoil. SETH MUSSER, Massachusetts Institute of Technology, DAVIDE PROMENT, University of East Anglia, MIGUEL ÓNORATO, University of Turin, WILLIAM IRVINE, University of Chicago — We investigate the development of superfluid flow around an airfoil accelerated to a finite velocity from rest. Using both simulations of the Gross-Pitaevskii equation and analytical calculations we find striking similarities to viscous flows: from the production of starting vortices to the convergence of the airfoil circulation onto a quantized version of the classical Kutta-Joukowski circulation. Using a phenomenological argument we predict the number of vortices nucleated by a given version of the classical Kutta-Joukowski circulation. Finally we analyze the lift and drag acting on the airfoil.

4:53PM B08.00002 Small particle motions in inhomogeneous super fluid turbulence. TAKUMI MARUYAMA, SHINICHIRO WAKI, SOU SUZUKI, VOLKER SONNENSCHEIN, HIDEKI TOMITA, YOSHIYUKI TSUJI, TETSUO IGUCHI, Nagoya University — Lagrangian trajectories of small particles in a fully developed turbulent state are studied in a rectangular duct. A plate heater is attached on the bottom to generate the thermal counter flow. The bath temperature is changed from 1.9 K to 2.1 K, and is controlled within 0.1 mK. Small He* particles of micron size and their trajectories are recorded. Their motions indicate complex features depending not only on bath temperature and heater power, but also on particle size. Particle motions near the wall are different from those at duct center. We report the effect of flow inhomogeneity on the particle motions. Smaller particle tends to be affected by the inhomogeneity than the larger particle sizes. To characterize the particle motions, the Hurst exponent is defined by $|\langle x(t+\tau) - x(t)\rangle| \propto \tau^\beta$, where $x(t)$ denotes the particle position at time $t$. It is found that there is a characteristic time scale $\tau_0$. For small time separation, $\tau \ll \tau_0$, the exponent $\beta$ is small. However, for large time separation, $\tau_0 \ll \tau$, $H$ is nearly $1/2$. [1] W.Kubo and Y.Tsui, Journal of Low Temperature Physics, 2010, Volume 196, pp 170176

5:06PM B08.00003 Quantum turbulence exploration using the Gross-Pitaevskii equation¹, LUMINITA DANAILA, University of Rouen Normandy, MICHIAKUZU KOYAYASHI, Kyoto University, FRANKLY LUDDENS, Corentin LOTHODE, University of Rouen Normandy, PHILIPPE D'ARNAUDEAU, University of Poitiers, IONUT DANAILA, University of Rouen Normandy, MARC BRACHET, Ecole Normale Superieure Paris, QUTE-HPC COLLABORATION — We solve numerically the Gross-Pitaevskii (GP) equation to simulate the dynamics of Quantum Turbulence (QT) in a periodic box. This intends to model the behaviour of superfluid helium in the low-temperature regime, therefore a viscous-free flow. Simulations are performed with a spectral code solving the GP equation using MPI-OpenMP parallel programming. We assess the effect of different initial conditions on the statistical behaviour of the flow, through both spectra and structure functions. Closures for non-linear energy transfer terms are proposed and validated, mainly based on vortex reconnection mechanism. Analogies and differences between QT and classical turbulence are drawn.

5:19PM B08.00004 A Laser system for flow field imaging in superfluid helium using He₂⁺. SO SUZUKI, VOLKER SONNENSCHEIN, TAKUMI MARUYAMA, SHINICHIRO WAKI, HIDEKI TOMITA, YOSHIYUKI TSUJI, TETSUO IGUCHI, Nagoya University — For visualization of the flow field in superfluid Helium (He II), He₂⁺ excimer can be traced by using Laser induced fluorescence (LIF). In our group the proposed He₂⁺ excimer generation method is based on the neutron absorption reaction of naturally abundant $^3$He in Helium. Small He₂⁺ clusters generated by this method are then available for 3D LIF imaging. By applying this method, we already confirmed successful generation of He₂⁺ excimers using our laser system. In this study, for successful visualization of He₂⁺ excimer images in superfluid Helium, the required laser parameters, such as f intensity, repetition rate and wavelength were investigated. For systematic characterization of these, an offline system based on a high voltage discharge for excitimer generation is being set up to provide a more intense and accessible source of excimers, not requiring the use of a neutron beam.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B09 Aerodynamics: Rotating Wing 213 - Bo Cheng, Penn State University

4:40PM B09.00001 Scaling arguments for the radial vorticity advection and the radial planetary vorticity tilting in the leading-edge vortex of revolving wings¹. NATHANIEL WERNER, Penn State University, JUNSHI WANG, HAIBO DONG, University of Virginia, BO CHENG, Penn State University — Within the leading-edge vortex (LEV) of a revolving wing, planetary vorticity tilting (PVTr) can partly remove the radial vorticity generated by advection, a mechanism that relates the effects of Coriolis acceleration, spanwise flow, and the tilting of the planetary vorticity. It has been shown previously that the non-dimensional PVTr scales independently with Aspect Ratio (AR) while the advection scales inversely with AR, suggesting that separate scalings should be applied to these two terms. This study continues the previous investigation, with the goal of finding the correct scaling of the radial vorticity advection with the AR. The wing AR is changed by increasing the wing span and keeping the wing chord length constant; in this process we either keep the wing tip velocity constant (i.e., keeping the wing-chord-based Reynolds number constant) or keep the wing angular velocity constant (i.e., keeping the spanwise local Reynolds number constant). For both cases, we show that the correct length scale for the advection is wing span instead of wing chord (as for the PVTr), while both the advection and the PVTr have the tip velocity and planetary vorticity (or wing angular velocity) as the velocity and vorticity scales.

¹NSF CMMI 1554429

4:53PM B09.00002 Hover Predictions Using a High-Order Discontinuous Galerkin Off-Body Discretization¹, KURSAT KARA, Oklahoma State University, MICHAEL BRAZELL, ANDREW KIRBY, University of Wyoming, EARL DUQUE, Intelligent Light, DIMITRI MAVRIPLIS, University of Wyoming — Hover performance of a four-bladed Sikorsky S-76 is studied using a high-order discontinuous Galerkin (DG) off-body discretization. Time accurate Navier-Stokes calculations are performed using the WAFKE3D code which combines solution technologies in a multi-mesh, multi-solver paradigm through a dynamic overset framework which employs NSU3D as a near-body solver and dg3est as an off-body solver. The rotor with swept-tapered tip is simulated. The tip Mach number was 0.65, and the Reynolds number based on the reference chord was 1.2 million. A constant coning angle of 3.5° is applied. Effect of time step size and sub-iterations on the integrated parameters are investigated. Convergence results are presented. The figure of merit is calculated and compared with available data in the literature, and good agreement is found.

¹Computer time was provided by the NCAR-Wyoming Supercomputer Center (NWSC) and the University of Wyoming Advanced Research Computing Center (ARCC).
5:06PM B09.00003 Numerical simulation of a rotating blade using a flat-plate airfoil at low Reynolds numbers for Mars helicopter

DAICHI OGASAWARA, MAKOTO SATO, Kogakuin University, HIDEAKI SUGAWARA, Ryosy Systems Co., Ltd., YASUTADA TANABE, Japan Aerospace Exploration Agency, KOTARO SATO, Kogakuin University — To realize a high-performance rotor for a Mars helicopter, the aerodynamic performances and flow characteristics around a rotating blade using a flat-plate airfoil at low Reynolds numbers have been investigated. Numerical simulations have been conducted using a CFD solver of rFlow3D, which has been developed in JAXA for a rotorcraft simulation. The ambient pressures are set to 100 kPa, 50 kPa, 10 kPa, and 5 kPa, corresponding to the Reynolds numbers of 77000, 39000, 7700, and 3900 based on the blade tip velocity and the tip chord length respectively. The thrust and torque coefficients agree well with the previous experimental study for the cases with pitch angles up to 10 deg. The influences of the ambient pressure on the aerodynamic characteristics are not so significant in the cases with low pitch angles. On the other hand, the flow characteristics become highly different between the cases with the ambient pressure of 100 kPa and 5 kPa.

5:19PM B09.00004 Scaling effects on aerodynamic interactions of rotorcraft around boundaries

DARIUS CARTER, MEGAN MAZZATENTA, SHIJIE GAO, CARMELO DI FRANCO, NICOLA BEZZO, DANIEL QUINN, University of Virginia, LINK LAB COLLABORATION — The growth of the Micro Aerial Vehicle (MAV) industry is outpacing our understanding of how MAVs behave in cluttered environments. Search and rescue and product delivery — two key MAV applications — occur in tight, confined spaces filled with complex obstacles. Our current understanding of how MAVs interact with boundaries is based primarily on helicopter models, which were designed for high Reynolds-number single-rotor flows. To support better flow models of MAV-boundary interactions, we will measure the lift forces and flow of small quadrotors near a side wall, ground, ceiling, and water surface. To see how our results scaled, we measured differently-sized propellers in these same conditions. Using Particle Image Velocimetry, we quantified the momentum flux of the rotors and evaluated the assumptions made by the existing ground and ceiling models. Better physical models offer a way to predict MAV’s reaction to environmental disturbances, which is critical for certifying that MAVs can operate safely near or in cooperation with humans. Better models could offer physics-based situational awareness, which could reduce the need for heavy sensors and cameras and free up payload on small, lightweight MAVs.

5:32PM B09.00005 Propellers in Partial Ground Effect

JIELONG CAI, SIDAARD GUNASEKRAN, University of Dayton, MICHAEL OL, Folderol, LLC, ANWAR AHMED, Auburn University — The classical result for propellers (or any rotating wing) operating parallel and in close proximity to an unbounded flat-plate, is increase in thrust and decrease in power-required, for a given rotation-rate. We examine extreme cases, where the ratio of ground-proximity to propeller-diameter is 0.1 or less. We also reverse the propeller direction, for a ceiling effect. The limiting case for small ground separation is halving of power-required. However, this depends strongly on the ratio of propeller pitch to diameter. For large ratios (approaching 1), ground-effect offers almost zero benefit. For small ratios (on the order of 0.5), best results are obtained. We also consider a finite ground-plane as a circular disk. For a ratio of disk diameter to propeller diameter of 0.5, ground-effect is nil, while for a ratio of 1, full ground-effect is restored. Flow visualization gives the explanation: there is a circular time-averaged dividing streamline on the ground-plane, within which the projection of the flow is swirling, but outside of which the flow is radial. For ground-plate-disks of diameter smaller than this dividing streamline, propeller thrust is measured to resemble that in free-air, while for larger disks there is a strong ground-effect.

5:45PM B09.00006 Investigating Effects of Cracked Blade under Various Rotor Speed on Aerodynamic Characteristics in 1.5 Stage GE-E3 Gas Turbine

THANH DAM MAI, MYUNG GON CHOI, YUMIN KIM, JAIYOUNG RYU, Department of Mechanical Engineering, Chung Ang University, Seoul, 06974, Republic of Korea — The variation in revolutions per minute (RPM) has significant effects on dynamic behaviors, the flow characteristics, and the heat transfer in gas turbine. Coupled with cracked blade conditions, it can be speculated to have more dramatic effects on the overall performance. This is the first study to address both RPM variation and cracked rotor blade condition in three-dimensional unsteady gas turbine simulation. Reynolds-averaged Navier-Stokes (RANS) equation is used with k-ω SST turbulence closure model to solve high-speed, high-pressure compressible flow in GE-E3 gas turbine. Numerical cases are selected with RPMs of 4000, 3600, 3200, and 2800 under same cracked blade conditions. As a result, average temperature and heat flux on the blade surface are found to have inverse relations with RPM. Pressure overshoot occurs at greater RPM changes, but significant drop in average temperature and heat flux is observed at 4000 RPM. Moreover, it is also found that crack condition has significant effects on the aerodynamic behavior of compressed gas in high pressure gas turbine.

5:58PM B09.00007 An Experimental Investigation on Drone Propellers Operating Under Icing Conditions

ZHE NING, YANG LIU, Iowa State University, HONGWEI MA, BeiHang University, HUI HU, Iowa State University — An experimental study was conducted to investigate the dynamic ice accretion process over the surfaces of typical drone propeller blades and to characterize the detrimental effects of ice accretion on the performance of the Drone propeller in terms of thrust generation and power consumption. The experimental study was conducted in the Icing Research Tunnel of Iowa State University (i.e., ISU-IRT) with a commonly-used, commercially-available Drone propeller exposed in frozen-cold incoming airflow under various icing conditions usually encountered by Drones in winters (i.e., under both rime and glaze icing conditions). In addition to revealing the transient ice accreting process over the surfaces of the rotating propeller blades by using a “phase-locked” imaging technique with a high-resolution imaging system, the dynamic thrust force generated by the Drone propeller was also measured simultaneously along with the required power inputs to drive the Drone propeller during the dynamic ice accreting process. The time-resolved aerodynamic force measurements and power consumption data were correlated with the acquired snapshots of the instantaneous ice accretion images to gain further insight into the underlying icing physics pertinent to Drone icing phenomena.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B10 Vortex Dynamics and Vortex Flow: Simulations

3A - Monika Nitsche, University of New Mexico
4:40PM B10.00001 Numerical study of polarized viscous vortex reconnection, JIE YAO, FAZLE HUSSAIN, Texas Tech University — Polarized vortical structures (i.e. with axial flow, hence coiled vortex lines) are generic to turbulent flows; hence studying their dynamics and interactions is essential to understanding turbulence phenomena. Vortex reconnection is a frequent event in turbulent shear flows and play an important role in energy cascade, mixing and noise generation. To quantify the polarization effect on vortex reconnection, direct numerical simulation of two anti-parallel helical vortex tubes is performed for vortex Reynolds numbers ($ \equiv \text{circulation/viscosity} \geq 9,000$ and initial swirl numbers $q \equiv \text{peak azimuthal velocity/peak axial velocity} \in [4, 0.75]$). For both the non-polarized and polarized cases, the reconnection event is delayed as polarization strength increases (i.e. $q$ decreases), but with a higher circulation transfer rate during reconnection. Compared with the unpolarized case, enstrophy and energy dissipation rates are suppressed for weaker polarized ($q=1.5$) cases, but surprisingly enhanced for strong polarized ($q=1.5$) cases. In addition, increasing the polarization strength alters the energy spectrum in the inertial range with a scaling varying from $k^{-5/3}$ for the unpolarized case to $k^{-7/3}$ for the strongly polarized cases. Hence polarization is found to significantly alter vortex reconnection dynamics.

4:53PM B10.00002 The Influence of Reynolds Number on the Dynamics of Wing-Tip Vortices, TOM SMITH, YIANNIS VENTIKOS, University College London — Tip vortex flows are of considerable interest across a range of technologies and can be significant noise sources, particularly in hydrodynamic applications where tip-vortex cavitation can occur. There are still many open questions regarding the dynamics of this type of flow including Reynolds scale effects and the dynamics of wake-like and horseshoe vortex interactions. In this study, we investigate the Taylor–Green vortex (TGV) and Direct Numerical Simulations to study the vortex roll-up process in laminar foils at low and moderate Reynolds numbers. Grid convergence is achieved to improve confidence in the accuracy of the simulations and to develop a better understanding of the mesh requirements for this type of flow. A detailed analysis of the roll-up process is carried out, highlighting significant differences in the flow as the Reynolds number increases. At low Reynolds numbers, a single laminar vortex forms with a very low axial velocity in the core. Higher Reynolds numbers see the emergence of multiple vortices which merge in the near wake. The magnitude and longitudinal position of the minimum pressure is also found to depend on the Reynolds number, which has important consequences for cavitation predictions and scaling.

5:06PM B10.00003 Formation and Evaluation of Vortex Rings from Saddle Shape Nozzle, VAHID SADRI, SHELLY SINGH-GRYZBON, ZHENGLUN WEI, AJIT P YOGANATHAN, Georgia Institute of Technology — The mitral valve (MV) is a complex structure with a saddle-shaped annulus, separating the left atrium and left ventricle (LV). During rapid filling of the LV, a three-dimensional asymmetrical vortex ring forms downstream of the annulus. In this study, we investigated the formation mechanism and evolution of vortex rings from saddle-shaped nozzles using direct numerical simulations. A normal human mitral annulus was used to make the saddle-shaped nozzle geometry. The flow was simulated at a jet Reynolds number of 2,000 (based on hydraulic diameter and jet velocity) with jet pulse length-to-hydraulic diameter ratio ($L/D_h$) ranging from 1 to 6. It was found that the vortex ring from saddle-shaped nozzles has an oscillatory deformation while propagating, which causes the ring to undergo axis-switching. Furthermore, ambient fluid entrainment during vortex ring formation feeds to the main vortex structure, causing circumferential flow that splits the main vortex into smaller portions around the circumference. Additionally, the vortex formation number calculated for low stroke ratio cases.

5:19PM B10.00004 Evolution of vortex-surface fields in the flow past a finite plate, WENWEN TONG, YUE YANG, College of Engineering, Peking University — We investigate the evolution of the vortex-surface field (VSF) in the three-dimensional flow past a finite plate at the Reynolds number of 300, aspect ratio of 2, and angle of attack of 30 degrees. The VSF method is extended to complex flows with an immersed boundary by adding a source term in the VSF evolution equation. The VSF iso-surfaces display that near-plate vortex surfaces first roll up from plate edges, and then form hairpin-like structures near the leading edge and semi-riblet structures near plate tips and in the wake. We quantitatively distinguish two types of vortical structures by null points of streamwise vorticity on VSF isosurfaces, and refer them to as the leading edge vortex (LEV) and tip vortex (TV). The VSF characterizes that the development of the LEV near tips is suppressed by the finite growth of TV. In the wake region, helical vortex lines are generated and their geometry and impulses are quantified based on the VSF isosurfaces for TV.

5:32PM B10.00005 Controlling vortex breakdown through heat addition and extraction, XIAO ZHANG, University of Maryland, JOSEPH CHUNG, University of Maryland, College Park, CAROLYN KAPLAN, University of Maryland, ELAINE ORAN, Texas A&M University — This work examines how heat addition and extraction can be used to control the modes of vortex breakdown. Three-dimensional, unsteady simulations of gaseous vortex breakdown were carried out by solving the unsteady Navier-Stokes equations. Different modes of vortex breakdown were obtained by matching swirl and Reynolds numbers from the literature. Then, heat was either extracted or added within the vortex core to quantify the influence of non-adiabatic conditions. The results show that a critical value of heat addition will force the double helix vortex to transition to a spiral and then to the double helix mode of breakdown. Heat addition with the double helix mode, however, will force the double helix to transition to a columnar vortex, entirely bypassing the spiral mode. We discuss these results and other findings related to the bubble mode of breakdown.

5:45PM B10.00006 Role of helicity in vortex breakdown, SAMEEN A, MANJUL SHARMA, Dept. of Aerospace Engg., Indian Institute of Technology Madras, Chennai 600036, India — We investigate the bubble-type vortex breakdown flow numerically with a model problem of flow inside a cylinder with top rotating lid (also known as Vogel-Escudier flow). The parameters of the flow are $Re = \omega R^2/\nu$ and aspect ratio $\Gamma = H/R$, depending on which flow exhibits steady or unsteady breakdown bubble topologies. We find that negative helicity density is generated by the rotating top lid and is injected in the bulk of the flow at a critical value of the Reynolds number in the event of vortex breakdown. The flow also shows two-dimensional-three-dimensional (2D–3D) characteristics for which the helicity density is decomposed into $r,z$-component ($h_{r,z}$) and out-of-the-plane component ($h_{\theta}$). We find that the topology of the breakdown bubble correlates directly to the decomposed helicity $h_{r,z}$. Using only the decomposed helicity, $h_{r,z}$, complete breakdown bubble is reconstructed for non-axisymmetric flows. This correlation indicates that the vortex breakdown is the interplay between the axial component of velocity and the azimuthal component of vorticity which are characterized by $h_{r,z}$.

5:58PM B10.00007 Accurate evaluation of near-singular integrals in vortex sheet and Stokes flow, MONIKA NITSCH, University of New Mexico — Inviscid vortex sheet or viscous Stokes flow can be efficiently computed using boundary integral formulations, which describe the fluid velocity by integrals over interfaces bounding fluid regions. For points off the interface, the velocity is given by singular integrals, whose computation is well understood. However, for points not on the interface, but near it, the velocity is given by near-singular which are difficult to compute. This talk presents a simple method to accurately compute these integrals. It is based on approximating the integrand by functions that capture the near-singularity and can be integrated exactly. The method is quite general and is applied here to planar vortex sheet and axisymmetric Stokes flow.
Cost of wall modeling for the two wall models will be discussed. Model are conducted. Accuracy of the WMLES calculations will be discussed in terms of prediction quality of key mean-flow statistics at two

\[ \text{Re} = 4000 \]

where a skewed mean velocity profile is generated as TBL along the floor of a square duct deflects through a 30° bend. The experiment of Schwarz & Bradshow (J. Fluid Mech. (1994), vol. 212, pp. 183-209) is considered, have not been assessed in such nonequilibrium flows, where some of the key modeling assumptions are deemed invalid (e.g., unidirectional flow, turbulent boundary layers (TBL) in nature and realistic geometries of engineering applications. Wall models for large-eddy simulation (LES) wall-resolved simulations of nonequilibrium turbulent boundary layers is constructed, including scenarios of adverse/favorable pressure gradient and heated/cooled walls. The non-equilibrium forcing is spatially diffuse, providing significantly large regions where equilibrium scaling laws cannot be reasonably assumed in the inner layer, and necessitating non-equilibrium models for inner-layer accuracy. A database of wall-modeled simulations is then constructed to assess the model ability to reproduce non-equilibrium effects in both the inner and outer flow. Several model classes are employed including ODE, algebraic, and dynamic slip variants. The favorable performance of several new non-equilibrium models of algebraic complexity (efficient and easy to implement) is assessed relative to the equilibrium models and benchmark wall-resolved simulations.

5:06PM B11.00003 Wall model for large eddy simulations accounting for particle effect. 1

This research was supported by the National Natural Science Foundation of China (Grants No. 11490551,) and the Fundamental Research Funds for the Central Universities (lzujbky-2016-k13, lzujbky-2018-k07).

1This work is supported in part by MEXT as a social and scientific priority issue to be tackled by using post-K computer. A part of this research uses computational resources of the K computer provided by the RIKEN Advanced Institute for Computational Science (project ID: hp150254, hp160205, hp170267, hp180185).

5:19PM B11.00004 Equilibrium/Non-Equilibrium Wall-Modeled LES of Airfoil Stall Phenomena at High-Reynolds Number

This is presented by the authors at the 51st AIAA Fluid Dynamics Conference, 23-26 June 2019, San Diego, CA, USA. The presented work is supported by the Japan Aerospace Exploration Agency (JAXA).

1This research was supported by NASA (80NSSC18M0155) and the Office of Naval Research (N00014-16-S-BA10).
5:45PM B11.00006 Assessment of Wall-Modeled LES for Turbulent Boundary Layers with Heated/Cooled Wall¹, RYO HIRAI, SOSHI KAWAI, Department of Aerospace Engineering, Tohoku University — In this talk, we first investigate the effects of wall heat flux induced by heated and cooled walls in zero-pressure-gradient flat-plate turbulent boundary layers by using wall-resolved large-eddy simulation (LES). Based on the wall-resolved LES database, we then assess the capability of the equilibrium wall model (Kawai and Larsson, PoF, 2012) for predicting the thermal turbulent boundary layers. By considering the behaviors of density and viscosity variations in the inner-layer of the thermal boundary-layer, one source of the major errors induced by the existing model is analyzed and a method that allows for the errors to be removed is proposed. Additionally, we also seek the possibility of further improvements in the wall model by modifying the mixing-length model based on the scaling laws derived from the analysis of the wall-resolved LES database.

¹This work was supported by Japan Society for the Promotion of Science KAKENHI Grant Number 18H01620. Computer time was provided by the K computer at the RIKEN Advanced Institute for Computational Science through the HPCI System Research project hp180158 and hp190121.

5:58PM B11.00007 A near-wall eddy viscosity for compressible turbulent flows based on velocity transformation with application to wall models, PRAHLADH S. IYER, National Institute of Aerospace, PEDRO S. VOLPIANI, JOHAN LARSSON, University of Maryland, College Park, SERGIO PIROZZOLI, Sapienza University of Rome, MUJEEB R. MALIK, NASA Langley Research Center — A near-wall eddy viscosity is derived for zero-pressure gradient compressible turbulent flows based on a velocity transformation kernel. This is only valid in the inner layer of the flow with application to wall models. A priori analysis is performed using high-speed Direct Numerical Simulation (DNS) databases of turbulent boundary layers and channels over a range of Mach numbers and thermal conditions using the new formulation to assess its accuracy, and compare it with existing eddy viscosity models. Preliminary results are encouraging, and indicate that the errors in the wall model prediction is directly correlated with the error in the velocity transformation. More specifically, the eddy viscosity obtained with the proposed methodology and using the recent compressible velocity transformation of Volpiani et al. (2019) yields accurate results for supersonic and hypersonic turbulent boundary layers.

Saturday, November 23, 2019 4:40PM - 5:58PM – Session B12 Separated Flows: Control 303 - Amy Lang, University of Alabama

4:40PM B12.00001 Reversing Flow Formation Induced by an Increasing Adverse Pressure Gradient in a Separating Laminar Boundary Layer¹, ANDREW BONACCI, AMY LANG, LEO SANTOS, University of Alabama — Reversing flow development in a laminar boundary layer is a determining factor in the development of dynamic stall. It has been demonstrated that shark scales are capable of being bristled by reversing flow leading to a passive mechanism for separation control. An investigation of the formation of the reversing flow over a smooth surface is key to understanding this passive flow-actuated bio-inspired separation control mechanism. A water tunnel facility is utilized to generate a separated region by using a rotating cylinder to induce an increasing unsteady adverse pressure gradient over a smooth flat plate. Growing the laminar boundary layer to a thickness of about 10 mm and Reynolds numbers of 1.67*10⁵ to 2.98*10⁵ makes the separation region more measurable with DPIV. Results are analyzed for reversing flow development in size and magnitude to better understand the viscous development of the separation point as a function of the strength and timescale of the increasing adverse pressure gradient.

¹US Army grant W911NF1510556 and NSF REU grant EEC 1659710

4:53PM B12.00002 Effect of perimetric suction at the rear of a square-back bluff body on the bi-stable dynamics of the wake flow¹, LUC PASTUR, ENSTA Institut Polytechnique de Paris, France, EN-CHI HSU, Institute of Mechanical Sciences and Industrial Applications ENSTA, Institut Polytechnique de Paris, VLADIMIR PAREZANOVIC, Khalifa University of Science and Technology, Abu Dhabi, UAE — Wake flows behind 3D bluff bodies are known to exhibit random switches between two mirror reflectional symmetry-breaking wake flows each associated with an increase of the drag. Consequently, it may be desirable to symmetrize the flow, if the symmetrization would consist in stabilizing the unstable symmetric flow state associated with a lower drag. One attempt in this direction may consist in steadily blowing symmetrically through perimetric slits at the rear of the body. However, it was recently shown that this mode of actuation could not symmetrize the flow. The present work is a preliminary investigation of the effect of a steady symmetric perimetric suction at the rear of the body, among other modes of actuation of the wake flow. For moderate flow rates, it is found that the transitions between the two states are promoted, indicating that the actuator has authority on the flow bistability. The resulting symmetrization of the probability distribution function of the pressure gradient is accompanied by a decrease of the base pressure, together with the slight reduction of the bubble length, as revealed by rear pressure and PIV measurements.

¹This work has been supported by the Khalifa University of Science and Technology under Award No(s). FSU-2018-21 and CIRA-2019-025.
²M. Grandemange et al, PRE 86, 035302 (2012).
³M. Lorite-Díez et al, to appear in PAMM.
5:06PM B12.00003 Control of the reflectional symmetry breaking mode of a square-back bluff body wake using a sweeping jet actuator\footnote{This work has been supported by the Khalifa University of Science and Technology under Award No(s). FSU-2018-21 and CIRA-2019-025.}, VLADIMIR PAREZANOVIC, ABDUL RAOUF TAJIK, Khalifa University of Science and Technology, LUC PASTUR, IMSIA-ENSTA Institut Polytechnique de Paris — The wake of a square-back 3D bluff body is dominated by a stochastic switching between two reflectional symmetry breaking wake flows\footnote{M. Grandemange et al., Phy. Rev. E 86, 035302}. The two mirror wakes are associated with an increased drag. We attempt to symmetrize the wake using a Sweeping Jet (SJW) actuator located at the top of the bluff body base. Arrays of SWJs have been used to reduce the drag of the 3D bluff body by reattaching the flow to an angled back\footnote{M. Metka and J. W. Gregory, J. Fluids Eng. 137 (5), 051108} or to trailing edge flaps\footnote{J. Schmidt et al., Exp. Fluids 56, 151}. In our study, only a single SJW is used, and its exit nozzle almost spans the entire width of the bluff body base. The SJW produces a jet which oscillates horizontally, along the same direction of the bistable wake switching. The sweeping motion of the jet is intended to interact with the wake state switches, which may lead to a symmetric wake. Base pressure measurements reveal a wake locked in a symmetric state in the horizontal plane, for a certain range of actuator mass flow rates. This result is accompanied by a change in the vertical orientation of the wake.

5:19PM B12.00004 Elimination of velocity deficit behind a cylinder using reinforcement learning\footnote{RGC of Hong Kong under GRF (Project No. 15214418) and The Hong Kong Polytechnic University (Project No. G-YBXXQ)}, FENG REN, HUI TANG, The Hong Kong Polytechnic University — In this study, we present an active flow control strategy through deep reinforcement learning (DRL) for eliminating the velocity deficit in the wake of a circular cylinder. A group of windward-suction-leeward-blowing actuators are adopted. Their individual velocities are automatically adjusted by the DRL agent through feedback signals from a downstream sensor array. Simulations are conducted using a GPU-accelerated lattice Boltzmann solver with multi-block mesh partition. The high-dimensionality and non-linearity features of this problem make it challenging to explicitly determine the control strategy. By adopting the DRL, the agent can learn from the time sequences of the sensors, actuators and a specified reward function through trials and errors, and finally converge and determine the optimal control strategy. Results show that the well-trained control strategy can eliminate 99.7% of the velocity deficit. The policy is further transferred for fluid-structure interaction situations, and results based on four representative cases show that the transferred control strategy is robust and can effectively eliminate the velocity deficit by around 96%. Overall, the current study offers an innovative view that could potentially help underwater vehicles achieve low detectability.

5:32PM B12.00005 DSMC Simulation of Flow Past a Circular Cylinder at Re = 100, J.R. TORCZYNSKI, M.A. GALLIS, Sandia National Laboratories — The Direct Simulation Monte Carlo (DSMC) method is used to simulate vortex shedding in the flow of a gas past a circular cylinder at Re = 100, Ma = 0.1 (incompressible), and Kn = 0.00162 (continuum). Sandia’s DSMC code SPARTA is used. The domain is 5 diameters upstream, 10 diameters downstream, and 5 diameters on both sides and is meshed with 0.72 billion square cells with sides of 1/4 mean free path. Each cell contains $10^4$ particles, for 72 billion particles total. The time step is 1/3 of the mean collision time. Simulations are run on Sequoia, an IBM Blue Gene/Q petascale supercomputer at LLNL. The Kármán vortex street arises without using any perturbation, and the lateral force on the cylinder is periodic with a Strouhal number of 0.175. Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-NA0003525. This paper describes objective technical results and analysis. Any subjective views or opinions that might be expressed in the paper do not necessarily represent the views of the U.S. Department of Energy or the United States Government.

5:45PM B12.00006 On The Flows Of Low-Aspect Ratio Rotating Circular Cylinders\footnote{Air Force Office of Scientific Research (AFOSR)}, HISHAM SHEHATA, ALBERT MEDINA, MATTHEW ROCKWOOD, Air Force Research Laboratory, Wright-Patterson AFB, OH — A low-aspect ratio circular cylinder is placed on a laminar boundary layer plate in a wall-normal orientation, and rotated about its respective axial coordinate. It is found that the rotary motion of the cylinder may disrupt the symmetry of mainstay vortices associated with a static low-aspect ratio pin, resulting in a dominant streamwise vortex formation. Such formation stems from asymmetry of the near body wake, induced by cylinder motion and downwash effects at the free-end of the cylinder. The influence of cylinder aspect ratio, cylinder height to boundary layer thickness, and rotational rate are explored in detail through volumetric reconstruction of the flow field via stereoscopic particle image velocimetry (SPIV).


4:40PM B13.00001 Magnetic Braking of Jovian Jet Flows, ASHNA AGGARWAL, UCLA, SUSANNE HORN, Coventry University, JONATHAN AURNOU, UCLA — The azimuthally-directed jet flows of the gas giants, Jupiter and Saturn, are amongst their most dominant surface features. Recent Juno gravity measurements have inferred that the zonal jets of Jupiter extend from the weather layer, where they are directly observed, down at least 3,000 km deep into the H-He molecular atmosphere. In addition, Jupiter’s electrical conductivity increases as the molecular envelope transitions to a liquid metal. As electrical conductivity increases, the strength of magnetic forces grows, acting as a resistive brake on the jet flows. We have developed a pseudo-spectral code that solves the Cartesian Navier-Stokes equations in 2-D with buoyancy and a quasi-static magnetic field to quantify the process of magnetic braking, thought to truncate the Jovian jets. We will present the results of a suite of direct numerical simulations (DNS) of shearing convection, similar to Goluskin et al., (J. Fluid Mech. 759, 360, 2014), where we vary the strength of an imposed transverse magnetic field. Depending on the value of the magnetic field, the jets are damped, strongly intermittent, or fully suppressed.
4:53PM B13.00002 The Elbert Subrange of Magnetostrophic Rotating Convection, SUSANNE HORN, Coventry University, JONATHAN AURNOU, University of California, Los Angeles — Classical linear stability analysis shows that convection subjected to rotation and a magnetic field is most easily excited when Coriolis and Lorentz forces are approximately in balance and in the form of a large-scale stationary bulk mode. Since estimates for Earth also suggest that the outer liquid metal core is in this so-called magnetostrophic state, there is a long-held belief that these modes optimise planetary magnetic field generation. But a single-mode theory is not likely to be geophysically realistic. Instead, liquid metal flows are characterised by pronounced multimodality with a mix of stationary, oscillatory, and boundary-attached modes. In fact, Donna Elbert (cf. Chandrasekhar, 1961) discovered that there is subrange of magnetostrophic rotating convection where two types of stationary modes co-exist: a small-scale geostrophic mode and a large-scale magnetostrophic mode. The parameter space for this subrange coincides with the one for planetary cores, suggesting a crucial link to the magnetic field generation in geo- and astrophysical objects. Here, we revisit linear stability results and further use nonlinear direct numerical simulations to verify which onset characteristics, such as length scales and frequencies, carry over to higher supercriticalities.

5:06PM B13.00003 Experimental study of the non-linear saturation of the elliptical instability: inertial wave turbulence versus geostrophic turbulence1, THOMAS LE REUN, BERNARD FAVIER, MICHAEL LE BARS, IRPHE, Aix-Marseille Universitat, ERC FLUDYCO TEAM — We present an experiment of the turbulent saturation of the flow driven by parametric resonance of inertial waves in a rotating fluid. An ellipsoid filled with water is brought to solid-body rotation and then undergoes harmonic modulation of its rotation rate. This triggers the exponential growth of a pair of inertial waves via the elliptical instability. As the instability reaches non-linear saturation, it creates a turbulence where energy is injected into the resonant waves only. Depending on the amplitude of the rotation rate modulation, two different saturation states are observed. At large forcing amplitudes, the saturation flow mainly consists of a steady, geostrophic anticyclone. Its amplitude vanishes as the forcing amplitude is decreased while remaining above the threshold of the elliptical instability. Below this secondary transition, the saturation flow is a superposition of inertial waves which are in weakly non-linear resonant interaction, a state that could asymptotically lead to inertial wave turbulence. The present study is an experimental confirmation of the model of Le Reun, PRL (2017) who introduced the possibility of these two turbulent regimes. The transition between these two regimes and their relevance to geophysical applications are finally discussed.

5:19PM B13.00004 How the Great Red Spot of Jupiter Stays Alive while Losing Energy through Viscous and Radiative Dissipation, AIDI ZHANG, PHILIP MARCUS, UC Berkeley — During the last decade, the cloud cover over the Great Red Spot (GRS) of Jupiter has shrunk significantly. This observation, along with recent observations that the GRS has been repeatedly shedding large (100,000 km$^2$) chunks of itself, has caused many planetary scientists to speculate that the GRS will vanish in the next 10 years. We argue against that hypothesis and demonstrate that GRS, which is a large anticyclone, maintains itself with a weak (and not directly observable) secondary circulation that is consistent with all of the observations. Numerical simulations of the anelastic ideal gas equations are used to show that this secondary circulation both re-energizes the GRS and creates new anti-cyclonic vorticity via baroclinic dynamics that can only exist in a vertically stably-stratified, rotating atmosphere. The secondary circulation brings energy from the atmosphere outside the GRS into its interior. The energy flux into the GRS balances the loss of energy of the GRS from viscous and radiative dissipation. The rate of viscous losses agrees with the rate of loss of stored angular momentum and thermal energy due to radiative damping. Without the secondary circulation of the GRS, radiative damping would cause the GRS to decay in 4-5 years.

5:32PM B13.00005 A quasi-geostrophic thermally driven model for rapid dynamics in Earth’s core, MEREDITH PLUMLEY, ANDREW JACKSON, ETH Zurich, STEFANO MAFFEI, University of Leeds — Simulating the physics of Earth’s core remains an elusive challenge that precludes numerical solution of the full governing equations. Model reduction must be embraced to study the dynamics occurring in the core. One alternative for rotating flows is to adopt a columnar approximation for the velocity field, sometimes referred to as the quasi-geostrophic approximation. This technique is used to derive a completely newmodel for the rapid dynamics in thermally driven convection on the equatorial plane by using axial averages. We show that the onset of thermal convection found using local theory applied to this model agrees quantitatively with the values found in full numerical investigations and asymptotic theory. Time evolution results from this model can provide insight into the dynamics on short timescales in the core. We investigate the scaling laws of supercritical convection in parameter regimes unachievable in 3-D models.

5:45PM B13.00006 Machine learning predictions for magnetic field time evolution in a Three-Meter liquid sodium spherical Couette experiment1, ARTUR PEREVALOV, RUBEN ROJAS, BRIAN HUNT, DANIEL LATHROP, University of Maryland College Park — The source of the Earth’s magnetic field is the turbulent flow of liquid metal in the outer core and its interaction with the present magnetic field. Our experiment’s goal is to create an Earth-like dynamo to explore the mechanisms of generating magnetic fields and to understand the involved dynamics. Full numerical prediction is a challenging problem as it is strongly nonlinear and not computationally feasible to resolve. We present the implementation of various prediction techniques, including a reservoir computer deep learning algorithm, to probe the feasibility of using magnetic field data from the experiment to predict future measurements. The experiment is a three-meter diameter outer sphere and a one-meter diameter inner core model with the gap filled with liquid sodium. The spheres can rotate independently up to 4 and 14 Hz respectively, giving a Reynolds number up to $1.5 \times 10^8$. Two external electromagnets apply magnetic fields, while an array of Hall sensors measure the resulting magnetic fields. We use this data to train a reservoir computer to predict Hall sensor measurements and mimic waves in the experiment. Surprisingly, accurate predictions can be made for several magnetic dipole time scales. This shows that such a MHD systems behavior can be predicted.

1 NSF EAR-1417148 EAR-1909055
Our experiments aim to provide a deeper understanding of the turbulent convection and its interaction with magnetic fields in turbulent low Prandtl number flows as those in molten steel, aluminum or in geo- and astrophysical flows.

1Deutsche Forschungsgemeinschaft GRK 1567 and No. VO 2332/1-1

4:53PM B14.00002 Mixing in magnetohydrodynamic turbulence1. X.M. ‘SHINE’ ZHAI, Numeca-USA, P. K. YEUNG, KIRANRAVIKUMAR, Georgia Institute of Technology — Turbulence is characterized by its ability to efficiently promote scalar mixing, which occurs at the molecular level as large scale non-uniformities are broken into ever smaller pieces accompanied by a classical energy cascade. Yet for electrically conducting fluid subject to a strong magnetic field, anisotropy develops in all scales of turbulence and energy cascade is inhibited (Zhai & Yeung, PRF 2018). As a result, scalar mixing in magnetohydrodynamic (MHD) turbulence is expected to show distinctly different behaviors. We have performed direct numerical simulations of scalar mixing in MHD turbulence under mean scalar gradient. When a magnetic field is present, mixing of scalars in liquid metals, which has a Schmidt number of $O(0.01)$, is weakened as scalar variances grow more slowly. Variances of scalar gradients grow faster for less diffusive scalars, but gradient fluctuations in the direction of the magnetic field are suppressed. The largest simulations for passive scalars were performed at a resolution of $8192 \times 2048^2$ on a domain size of $32\pi \times (4\pi)^2$. The case of active scalars is also briefly addressed.

1National Science Foundation, Texas Advanced Computing Center

5:06PM B14.00003 Decay of turbulence in a duct with transverse magnetic field

OLEG ZIKANOV, University of Michigan - Dearborn, DMITRY KRASNOV, THOMAS BOECK, Ilmenau University of Technology, SEMION SUKORIANSKY, Ben-Gurion University of the Negev — Decay of honeycomb-generated turbulence in a duct with a static transverse magnetic field is studied via high-resolution direct numerical simulations. The simulations follow the experimental study of Sukoriansky et al., 1986, in particular the paradoxical observation of high-amplitude velocity fluctuations, which exist in the downstream portion of the flow when the strong transverse magnetic field is imposed in the entire duct including the honeycomb exit, but not in other configurations. It is shown that the fluctuations are caused by the large-scale quasi-two-dimensional structures forming in the flow at the initial stages of the decay and surviving the magnetic suppression. Statistical turbulence properties, such as the energy decay curves, two-point correlations and typical length scales are computed. The study demonstrates that turbulence decay in the presence of a magnetic field is a complex phenomenon critically depending on the state of the flow at the moment the field is introduced.

5:19PM B14.00004 Laboratory Measurement of Non-Rotating Magnetooconvection in Liquid Gallium: Wall-mode Onset and Supercritical Precessional Mode1. YUFAN XI, University of California, Los Angeles, SUSANNE HORN, Coventry University, JONATHAN AURNOU, University of California, Los Angeles — Magnetooconvection in the Earth’s molten outer core, driven by convection, generate a planetary-scale, nearly axial, dipole-dominated magnetic field. The behaviors of strongly turbulent convection in the presence of strong Lorentz forces are mostly unknown. Thus, we present results of laboratory experiments on non-rotating Rayleigh-Bénard convection of liquid gallium in the presence of a vertical magnetic field. Our heat transfer survey in a diameter-to-height aspect ratio $\Gamma = 1$ tank, with $10^6 < Ra < 10^9$ and $0 < Ch < 3 \times 10^9$, shows that the convection onset well below the predicted $Ra$ for an infinite fluid layer. Magnetooconvection in our finite cylindrical tanks likely onsets via stationery wall-attached modes (Houchens et al., 2002; Busse, 2008). In a $\Gamma = 2$ tank and with $Ra \approx 2 \times 10^9$, we vary the applied magnetic field corresponding to interaction parameter numbers $N$ from 0 to 10. Our thermal measurements show the existence of a novel precessional mode at $N \approx 0.5$ with electrically conducting boundaries (Cu), but not with electrically insulated boundaries (Teflon coated Al). This finding suggests the possibility of slowly traveling magneto-precessional modes attached to the electrically conductive boundaries of Earth’s outer core.

1NSF EAR
literature. A theoretical model is being sought to represent this interaction and predict its effect on the near field entrainment. 

Using conditional averaging, this organization was distinguished in the ring and braid region. For the excited jets, it agreed well with the organization of streamwise vortices from radial (in unforced case) to an azimuthal configuration that has been classically observed at low particle image velocimetry was performed at one and two diameters downstream of the nozzle. We varied the strengths of the axisymmetric where drop formation happens. In most views on jet shape and formation the role of surface tension is neglected; our experiments however demonstrate that both the ballistic and tip region of the jet are strongly decelerated by surface tension forces, that also affect the jet shape. 

Several theories exist relating jet shape and internal fluid motion, but none so far can describe all beautiful shapes remain incompletely understood. We perform PIV experiments to probe dynamic fluid motion inside an emanating jet to relate the fluid flow to the jet evolution and shape. 


demonstrate that both the ballistic and tip region of the jet are strongly decelerated by surface tension forces, that also affect the jet shape. Several theories exist relating jet shape and internal fluid motion, but none so far can describe all the different features of the jet. We find that for some experimental parameters that the jet acquires a large upward velocity in a small upward acceleration region located near the jet base due to collapse of a cavity formed during impact of the falling drop. The PV allows to distinguish three jet regions: a small acceleration region, a long ballistic region where fluid moves with a nearly constant momentum, and a jet tip region where drop formation happens. In most views on jet shape and formation the role of surface tension is neglected; our experiments however demonstrate that both the ballistic and tip region of the jet are strongly decelerated by surface tension forces, that also affect the jet shape. 

4:40PM B15.00001 Particle image velocimetry inside emanating jets to study jet shape and evolution. CEES VAN RIJN, DANIEL BONN. University of Amsterdam, BODJIE VAN BRUMMEN, JERRY WESTERWEEL. Technical University Delft, FLUID MECHANICS GROUP TUD COLLABORATION. — Jets resulting from drop impact on a fluid are a fascinating and remarkably robust phenomenon. However, their beautiful shapes remain incompletely understood. We perform PIV experiments to probe dynamic fluid motion inside an emanating jet to relate the fluid flow to the jet evolution and shape. Several theories exist relating jet shape and internal fluid motion, but none so far can describe all the different features of the jet. We find that for some experimental parameters that the jet acquires a large upward velocity in a small upward acceleration region located near the jet base due to collapse of a cavity formed during impact of the falling drop. The PV allows to distinguish three jet regions: a small acceleration region, a long ballistic region where fluid moves with a nearly constant momentum, and a jet tip region where drop formation happens. In most views on jet shape and formation the role of surface tension is neglected; our experiments however demonstrate that both the ballistic and tip region of the jet are strongly decelerated by surface tension forces, that also affect the jet shape. 

4:53PM B15.00002 A Tensor Formulation of the Lumley Decomposition Applied to the Jet Far-field. AZUR HODZIC, KNUD ERIK MEYER, Technical University of Denmark, WILLIAM K. GEORGE, Imperial College London, CLARA M. VELTE, Technical University of Denmark — A tensor formulation of the Lumley Decomposition (LD) is applied to the jet far-field which is analyzed in curvilinear coordinates. Fourier-based eigenfunctions along the streamwise direction are deduced directly from the LD, but only if a specific weighted inner-product space is chosen for the decomposition. A Galerkin projection of the turbulence kinetic energy (TKE) production term is then analyzed in order to quantify the TKE production capabilities of the eigenfunctions. It is shown that a wide range of mode- and wavenumbers exhibit similar rates of energy extraction directly from the mean flow, as hypothesized by [Winstrm 2009]. Data obtained from PIV measurements of the jet far-field then suggest that the energy transport between eigenfunctions is not strictly governed by the classical Richardson cascade model, but that a wide range of modes are at least partially able to circumvent this cascade. [Wänström 2009] Spatial decompositions of a fully-developed turbulent round jet sampled with particle image velocimetry. Ph.d. thesis, Chalmers University of Technology. 

This project has received funding from the European Research Council (ERC) under the European Unions Horizon 2020 research and innovation program (grant agreement No 803419). 

5:06PM B15.00003 The dynamics of streamwise vortices in high Re jets. JAHNAVI KANTHARAJU, DAAA, ONERA Meudon 92190, BENJAMIN LECLAIRE, LAURENT JACQUIN, DAAA, ONERA Meudon 92190 France — Streamwise vortices play an important role in the mixing and entrainment in the near field of jets. They are dynamically active structures that lead to additional flow features such as side jets and streaks. Their coexistence with and wrapping around the vortex rings suggest their possible influence on the rings as has been remarked in several studies. We present results supporting one such possible interaction between streamwise vortices and the axisymmetric \(\text{(} n = 0 \text{)}\) mode in round jets (Davoust et. al JFM 2012) at \(Re\) in the range of \(9.2 \times 10^3\) to \(3.5 \times 10^4\). High speed particle image velocimetry was performed at one and two diameters downstream of the nozzle. We varied the strengths of the axisymmetric mode relative to streamwise vortices through acoustic excitation. It is shown that as the axisymmetric mode is strengthened, there is a shift in the organization of streamwise vortices from radial (in unforced case) to an azimuthal configuration that has been classically observed at low \(Re\). Using conditional averaging, this organization was distinguished in the ring and braid region. For the excited jets, it agreed well with the literature. A theoretical model is being sought to represent this interaction and predict its effect on the near field entrainment.
5:19PM B15.00004 Three-dimensional Lagrangian statistics of a turbulent free-shear jet, BIANCA VIGGIANNO, Portland State University, THOMAS BASSET, ROMAIN VOLK, LAURENT CHEVILLARD, MICKAËL BOURGON, Laboratoire de Physique de l’ENS de Lyon, Raul BAYON CAL, Portland State University — A fundamental study of a free-shear jet is performed to investigate the highly anisotropic and inhomogeneous flow field through Lagrangian statistics. Experiments were conducted within a Lagrangian Exploration Module, an icosahedron apparatus, to facilitate optical access of three high speed cameras. This enables the generation of 3D particle trajectories by implementing stereoscopic particle tracking velocimetry and results in three component tracks of position, velocity and acceleration of the tracer particles within the vertically-oriented jet. The total measurement volume is 80 x 80 x 200 mm³, which includes over 50 nozzle diameters downstream, where the jet becomes self-similar. Lagrangian statistical analysis is performed to identify intermittent behavior at the centerline of the jet and at the turbulent/non-turbulent interface. Further, the applicability of Taylor’s frozen flow hypothesis is investigated.

5:32PM B15.00005 Non-equilibrium Turbulent/non Turbulent Interface velocity scaling in turbulent planar jets, GIOACCCHINO CAFIERO, Politecnico di Torino, JOHN CHRISTOS VASSIDILICOS, Imperial College London — We investigate the turbulent/non-turbulent interface (TNTI) of a turbulent planar jet using stereo-PIV and HWA. Following Zhou & Vassisilos (JFM 2017) we derive an expression for the entrainment velocity \( \frac{\nu_n}{\nu} \), where \( \nu \) is the streamwise coordinate, \( \nu \) is the streamwise velocity, \( A_T \) is the area of the turbulent region in a plane orthogonal to the mean flow, and \( L \) is the length of the TNTI in that plane. \( \nu_n \) can be expressed as \( \nu_n \sim L^{-\eta} \) where \( \eta \) is the jet width at \( x \). Our experimental results confirm Corrsin’s \( \eta \sim \nu/v \) (Corrsin 1955) is a characteristic interface thickness and \( \eta \) is the line interface’s fractal dimension. Non-equilibrium dissipation self-similar jet theory (Cafiero & Vassilicos PRSA 2019) predicts that \( \nu_n/\nu \) (where \( \nu \) is the Kolmogorov velocity) is a decreasing function of \( x \), which is at odds with the classical Corrsin scaling \( \nu_n \sim \nu \). Our experimental results confirm Corrsin’s \( \eta \sim \nu/v \) but also show that \( \nu_n/\nu \) is indeed a decreasing function of \( x \). Our measurements support the scaling \( \nu_n \sim \nu \) (\( \nu \) being \( \lambda \) the Taylor scale) predicted by our theory.

5:45PM B15.00006 Round and Plane, Turbulent, Twin Jets, TAYE MELAKU TADDESSE, JOSEPH MATHEW, Indian Institute of Science — Multiple jets are found in several applications such as fuel injection, smoke stacks, jets engines of aerospace vehicles, etc. Even a pair of round jets can have a distinctive development because, after the inevitable merging, there can be axis switching. While plane jet pairs may be expected to behave like a single plane jet after merger, significant differences have been observed in experiments. LES of parallel twin jets, round and plane, reveal the structure and mechanisms of these two types. The LES is by an explicit filtering method. The twin round jets are at a Reynolds number Re = 2.3 x 10⁵, based on diameter and mean velocity at exit. Distance between jet centers was 5 diameters. Close quantitative agreement with experiment was found on the development of mean profiles and spreading. Velocity fluctuations between the jets are weaker than those on the outer boundaries. Axis switching was observed. Twin plane jets were at \( Re = 8.75 \times 10^4 \) based on jet width, spaced 12 jet widths apart were also simulated. The enhanced growth rate of twin jets has been captured in the simulations.

5:58PM B15.00007 Mixing of temperature and helium in turbulent co-axial jets¹, ALAIS HEWES, LAURENT MYDLARSKI, McGill University — There are relatively few studies of turbulent multi-scalar mixing, despite the occurrence of this phenomenon in common processes (e.g. chemically reacting flows, oceanic mixing). In the present work, we study the mixing of two passive scalars (temperature and helium concentration) in turbulent co-axial jets using a novel, 3-wire, thermal-anemometry-based probe designed to simultaneously measure velocity, temperature, and concentration. We emphasize that unlike most previous investigations of multi-scalar mixing, the instantaneous velocity field is measured in addition to the scalar fields, as it is required to fully describe turbulent scalar mixing. Our experiments are performed in vertically oriented co-axial jets consisting of a central jet of helium and an annular flow of (unheated) pure air, emanating into a slow co-flow of (pure) heated air (similar to the experimental set-up of Cai et al. (J. Fluid Mech., 2011)). We present measurements made in the near field of the jets, including variances and scalar fluxes, and focus on the evolution of the flow of (unheated) pure air, emanating into a slow co-flow of (pure) heated air (similar to the experimental set-up of Cai et al. (J. Fluid Mech., 2011)). We present measurements made in the near field of the jets, including variances and scalar fluxes, and focus on the evolution of the flow of (unheated) pure air, emanating into a slow co-flow of (pure) heated air (similar to the experimental set-up of Cai et al. (J. Fluid Mech., 2011)).

¹Graciously funded by NSERC.

Saturday, November 23, 2019 4:40PM - 6:11PM — Session B16 Advances in CFD Algorithms I 4c3 - Rajeev Jaiman, UBC

4:04PM B16.00001 Intersection-Based ALE for Radiation Hydrodynamics, PATRICK PAYNE, MARC CHAREST, HYEONGKAE PARK, Los Alamos National Laboratory — Many of the existing methods for Lagrangian hydrodynamics utilize staggered-grid hydrodynamic (SGH) algorithms. However, the fact that the momentum is defined by a nodal- and cell-centered coordinate,

4:53PM B16.00002 Robust data assimilation using mixed-norm optimization, SOUVIK GHOSH, Imperial College London, VINCENT MONSE, OLIIVIER MARQUET, DENIS SIPP, ONERA, France, PETER SCHMID, Imperial College London — Experimental data are often contaminated with outliers which in turn influence the quality of recovery in data assimilation techniques. We develop and present a computational framework based on mixed-norm optimization to determine flow fields from experimental measurements via a data-assimilation technique. More specifically, we use a variational adjoint-based methodology to balance a recovery error with a sparsity constraint, resulting in a saddle-point problem. The method shows promise in situations where only sparse measurements are available. Applications from mean-flow recovery at lower Reynolds numbers, as well as Reynolds-stress recovery at higher Reynolds numbers, will be presented.
This work was supported by the Center for Autonomous Systems and Technologies (CAST).
4:53PM B17.00002 Modeling rotor wakes of a quadrotor UAV in hovering mode\textsuperscript{1}, SEUNGCHEOL LEE, HUNGSON JOO, JOOHA KIM, Ulsan National Institute of Science and Technology, UNIST, Korea — For rotary-wing aircraft, predicting the locations of the rotor-tip vortices plays an important role in determining the rotor performance. Thus, there have been various attempts to model the rotor wake geometry for a single rotor. However, when these models are applied to multi-rotor UAVs, the interaction between rotor wakes, such as wake deflection, cannot be modeled. In the present study, we develop an empirical model that can predict the wake geometry for a quadrotor UAV in hover. The experiment is performed in a chamber at Re = 34,000, where Re is the Reynolds number based on the rotor chord length and the rotor-tip speed. By varying the normalized rotor separation distance (d/D) from 0.06 to 1.18, we directly measure the thrust force and the velocity field in the rotor wake using PIV, where d is the distance between adjacent rotor tips and D is the rotor diameter. With decreasing the normalized rotor separation distance, the extent of rotor-rotor interaction increases, and thus the wake center moves more to the center of UAV as the wake develops downstream. The wake geometry is predicted by modeling the locations of the wake center and the wake diameter as functions of the normalized rotor separation distance. Details will be discussed in the presentation.

\textsuperscript{1}Supported by NRF (2016R1D1A1B03933176, 2019R1F1A1064066), Research Fund (1.190011.01) of UNIST and Civil-Military Technology Cooperation Program (18-CM-AS-22).

5:06PM B17.00003 Characterization of Rotor-Rotor Aerodynamic Interactions for Free Flight Studies of Multirotor Systems\textsuperscript{1}, MARCEL VEISMANN, MORTEZA GHARIB, Caltech — As multirotor systems are increasingly utilized for academic as well as commercial purposes, their aerodynamics require precise identification for performance evaluation and more reliable predictive models. However, the interactions between closely arranged rotors have been given little attention. In order to account for the interactional effects on generated thrust, we developed an analytic expression derived from experimental data that is dependent on geometric and operational parameters, specifically rotor separation, rotational speeds, and Reynolds number. We find that the thrust reduction caused by the flow of neighboring rotors must be considered to accurately predict a multirotor system’s cumulative propulsive force. Using this analytical model, we aim to quantify rotor performance in various free flight conditions.

\textsuperscript{1}The presented material is based upon work supported by the Center for Autonomous Systems and Technologies (CAST) at the California Institute of Technology

5:19PM B17.00004 Experimental Measurement of Flow Field Around a Rotary Wing Unmanned Aircraft for Evaluation of Onboard Anemometer Placement\textsuperscript{1}, AYLSSA AVERY, SEABROOK WHYTE, JAMES BRENNER, VICTORIA NATALIE, JAMEY JACOB, Oklahoma State University — CLOUD-MAP is a collaboration between multiple universities to develop and evaluate unmanned aircraft technologies to place atmospheric sensors in otherwise hard to reach areas in the atmosphere. To determine the effectiveness of ultrasonic anemometers mounted on rotary wing aircraft, multirotor aircraft have been outfitted with sonic anemometers to measure windspeed and direction of winds aloft. The experiment maps the in-flow around a multirotor using multiple diagnostics, including PIV, ultrasonic anemometers, multi-hole probes, and impeller anemometers. Evaluation begins with flow measurements around a fixed single multirotor arm and followed up with multiple rotors with increasing fidelity for in-flight UAS. Multirotors are also placed in a wind tunnel and angled to simulate a sonic anemometer crosswind measurement. Comparisons with flight modes are made utilizing the OSU Gust and Shear Wind Tunnel Areas while validation tests are performed with tower mounted anemometers both at the Oklahoma Mesonet and the DOE ARM SGP Site. Areas of general quiescent flow are found absent of large scale disturbances or turbulent fluctuations. Results detail the accuracy of sonic anemometer measurements from mounted while on the multirotor in flight.

\textsuperscript{1}Supported by National Science Foundation Grant No. 1539070

5:32PM B17.00005 Flight performance of autonomous quadrotors in von Karman wakes, KASEY LAURENT, JOSHUA SOLBERG, MARIGOT FACKENTHAL, GRACE DING, GREGORY BEWLEY, Cornell University — Natural environments are highly variable and present significant challenges for autonomous aircraft. As the size of aircraft decreases, they become more susceptible to flow disturbances and typical control schemes fail. An understanding of the effects of turbulence on flight becomes crucial when developing new control schemes to optimize the performance of smaller aircraft. We designed an experiment in which an autonomous controller flies a small quadrotor in the von Karman vortex street behind a cylinder in a wind tunnel. We varied controller parameters as well as properties of the flow, such as mean airspeeds up to 3 m/s and wake dimensions as well as flow time scales between 0.5 and 2 times the quadrotor dimensions and response times, to find relationships between the controller and the response of the quadrotor. We measured displacements and power consumption in flight in the wake and compared this to the performance of the quadrotor in quiescent flow. Finally, we compare to the behavior of the quadrotor in active grid turbulence, with turbulence intensities up to 26%. We find that the quadrotor dynamics are set by an interplay between the turbulence and the controller.

5:45PM B17.00006 Potential Flow Model for Wall Effects on Quadrotor UAVs\textsuperscript{1}, DMITRI CALOMIRIS, JOVAN NEDIC, McGill University — Solid surfaces are known to directly influence the flow surrounding propellers when they are close to each other by altering the thrust force. One of the most common potential flow models, developed by Cheeseman and Bennett, predicts the relative increase of a rotor’s thrust as a function of ground distance (for constant power operation), within four rotor radii (i.e. in ground effect). Recently, Sanchez-Cuevas et al. extended this single propeller model to a quadrotor, which experiences a much larger influence from the ground. However, no such characterization has been done for walls. A potential flow model to account for wall effects will be presented and the results compared to experimental data. Initial measurements have qualitatively shown that the presence of the wall causes the quadrotor UAV to pitch down, essentially flipping it into the wall.

\textsuperscript{1}This work is partially funded by the FRQNT BIX Scholarship

5:58PM B17.00007 Multi-rotor Unmanned Aerial Vehicle (UAV) performance under shear flow\textsuperscript{1}, NINGSHEAN WANG, ALEKSANDAR DZODIC, DOM DIDOMINIC III, AMIT SANYAL, MARK GLAUSER, Syracuse University — Several multi-rotor UAV test flights under shear flow turbulence are carried out. This research targets evaluating the flight performance of the multi-rotor UAV when exposed to shear flow. The UAV deployed with different types of control schemes is exposed to shear flow with different shear rates and freestream velocities to gain its performance under a variety of conditions. Characterization of the shear flow field is evaluated by an array of total pressure scanners to measure the shear rate and free stream velocity spatially. The flight performance data is achieved through the Inertial Measurement Unit (IMU) integrated inside the UAV controller.

\textsuperscript{1}The authors are grateful to the support from Syracuse University and Dr. Mark Glauser for this research.
behavior of alignments are unaffected by the grid-rotation protocols and the modest transverse inhomogeneity. We also find that the local vorticity vector aligns with the large-scale vortical structures being stretched and aligned with the mean strain rate. The results show that the flow characteristics of the circular cylinder strongly depends on the Reynolds number. As the Re increases, this size and shape of wake structures become smaller and narrower with the dramatic reduction of the drag. This is due to the one and two bubble transition leading to the reattachment and delay of separation point. Although the general flow and load characteristics of the circular cylinder are well documented in the literature, it is still hard to find the detailed information of flow fields and local pressure distributions on the circular cylinder, especially at the supercritical Reynolds number. Therefore, this study will be able to provide meaningful information to improve our understanding of the physical insight into the relationship between the loads and wake structures. Moreover, this will contribute to the validation of CFD codes.

This research was supported by a grant from Endowment Project of Research of Turbulent Flow around Circular Cylinder using 3D PIV and Comparative Study of Turbulent Modeling funded by Korea Research Institute of Ships and Ocean engineering (PES 3320).

Measurement of flow fields and local pressures on a circular cylinder at the Reynolds numbers up to $10^6$. JAE HWAN JUNG, SEOKKYU CHO, Korea Res Inst of Ships and Ocean Eng (KRIOS), HYUNWOO JUNG, Seoul Natl Univ, JUN-HEE LEE, YONG-GUK LEE, HONG GUN SUNG, BU-GEUN PAIK, Korea Res Inst of Ships and Ocean Eng (KRIOS) — The present study is aimed to measure the turbulent flow fields and local pressures on a circular cylinder at the Reynolds numbers up to $10^6$. Unlike the previous studies mainly focused on the subcritical flow regime, the supercritical flow regime, which is more realistic to the full-scale condition of engineering, is considered in the present study. The particle image velocimetry (PIV) and local pressure scanner systems are employed to measure local mean pressures and flow fields on the circular cylinder. The results show that the flow characteristics of the circular cylinder strongly depends on the Reynolds number. As the Re increases, this size and shape of wake structures become smaller and narrower with the dramatic reduction of the drag. This is due to the one and two bubble transition leading to the reattachment and delay of separation point. Although the general flow and load characteristics of the circular cylinder are well documented in the literature, it is still hard to find the detailed information of flow fields and local pressure distributions on the circular cylinder, especially at the supercritical Reynolds number. Therefore, this study will be able to provide meaningful information to improve our understanding of the physical insight into the relationship between the loads and wake structures. Moreover, this will contribute to the validation of CFD codes.

5:06 PM B18.00003 Turbulent characterization generated by an active grid in a circular geometry. LOC DUFFO, R. JASON HEARST, Norwegian University of Science and Technology, THIERRY SCHULLER, Institut de Mcanique des Fluides de Toulouse, JAMES R. DAWSON, Norwegian University of Science and Technology — Experimentally inducing turbulence is of importance in understanding many practical flows. Recently active grids have been used to control the turbulence intensity and the integral length scale. However, these studies are almost exclusively based on diamond grids inserted in a square section. The aim of the current study is to design and test an active grid suitable for small diameter circular sections, with the ultimate goal of controlling inlet conditions for reacting flows. The characterization of the turbulence created by the circular sector paddles include various parameters such as paddle rotations speeds, number and shapes of paddles and number of active grid stages. These parameters were tested and compared in order to find the most efficient way to obtain a high level of turbulence intensity while retaining homogeneity. High turbulence intensities in the range of 10 to 20% were achieved using the circular geometry, which was comparable to that of previous square geometry studies. These experiments and results will enable the study of high intensity turbulence in circular geometry.

5:19 PM B18.00004 Influence of Atmospheric Flow Conditions on Scalar-mediated Mosquito Behavior. YI-CHUN HUANG, Princeton University, NEIL VICKERS, University of Utah, MARCUS HALTMARK, Princeton University — Mosquito-vectored diseases are increasingly problematic around the globe where they pose a current and emerging threat to public health. Female mosquitoes locate potential hosts by tracking CO2, volatile skin emanations, humidity, and thermal cues, each of which act as passive scalars that are distributed by local flow conditions. In a preliminary study, 3D field measurements of wind conditions and simultaneous mosquito captures in CO2-baited traps have shown that flight activity occurs during turbulent conditions. However, using field data, it is impossible to uncouple turbulent statistics from each other and from the mean wind speed. To fully explore the parameter space, a unique active flow modulation grid of independently operated paddles was developed. Unlike static grids that generate turbulence within a predefined range of scales, an active grid imposes variable and controllable turbulent structures onto the moving air by synchronized rotation of the paddles at specified frequencies. In the long-term, by leveraging such technology, host-seeking orientation strategies of these insects can be studied and the efficacy of traditional or new approaches that target the behavioral responses of mosquitoes can be rigorously assessed.

5:32 PM B18.00005 Turbulence scaling in the heliosheath and in the local interstellar medium from Voyager observations. FEDERICO FRATERNELLE, Politecnico di Torino, NIKOLAIV. POLOGORELOV, University of Alabama in Huntsville, JOHN D. RICHARDSON, MIT, DANIELA TORDELLA, Politecnico di Torino — This study considers the recent observations of magnetic field fluctuations in the inner heliosheath (IHS) and in the Local Interstellar Medium (LISM). We analyze in-situ data provided by the Voyager In- terstellar Mission (V1 and V2 spacecraft) for heliocentric distances up to 115 AU (V2) and 135 AU (V1). Turbulence, instabilities and recon- nection near the heliopause affect the particle transport and magnetic energy conversion into kinetic energy and into heat. A relevant medi-ation effect on plasma fluctuations is expected from charge exchange processes, which extensively generate pickup ions and energetic neu- tral atoms. We discuss the multi-scale features of magnetic fluctuations for a six-decades range of spacecraft-frame frequencies, which includes the energy-injection regime, the inertial cascade and the fluid-to-kinetic transition. In the IHS, our study reveals the presence of two relevant scales within the magnetohydrodynamic regime, and highlights a cor- relation between the intermittency and compressibility of fluctuations. In the LISM, we show that the dominant Alfvnic character observed in 2015/2016 is not retained during 2017, when the large-scale compressibility reaches the value of 0.6.
fluctuations in warm cloud and clear air environments. A Lagrangian set of data can then be obtained by fusion of GPS (Global Positioning System) and IMU (Inertial Measurement Unit) sensor output. GPS and IMU data can be integrated using the Kalman filter, where GPS provides the periodic updates for removing drifts in the IMU output. This data set can be of help in developing stochastic models to account for turbulence effects on cloud formation.

Saturday, November 23, 2019 4:40PM - 6:11PM
Session B19 CFD: Advanced Methods and Models 1

4:40PM B19.00001 In situ data compression for large-scale computational fluid dynamics simulations via interpolative decomposition methods1, HEATHER PACELLA, Stanford University, ALEC DUNTON, ALIREZA DOOSTAN, University of Colorado, Boulder, GIANNI ELIAC, Stanford University — Over the next decade, exascale supercomputers will provide a thousand-fold increase in floating-point performance, with memory increase of only a factor of one hundred. As a result, data generation from computational fluid dynamics simulations will easily surpass available memory capacity. Additional data compression techniques and optimization will also contribute to this discrepancy. To address this, we implement a lossy in situ compressive algorithm, interpolative decomposition (ID), within the solvers themselves, which allows us to store simulation results at a fraction of memory cost. Because ID algorithms operate independently on subregions of the fluid domain, they are a natural fit for the flexibility that task-based parallelism programming systems provide. Legion is one such programming system; it allows for implicitly extracted parallelism, easy performance tuning, and porting to various heterogeneous architectures. We will discuss the implementation of the interpolation and optimization within Legion, as well as performance, scalability, and cost of both ID algorithms in a task-parallel environment.

5:06PM B19.00003 Modeling the Effect of Resolution Inhomogeneity in LES, GOPAL YALLA, ROBERT MOSER, TODD OLIVER, Oden Institute for Computational Engineering and Sciences, The University of Texas at Austin, SIGFRIED HAERING, Argonne National Laboratory, BJORNE ENQUIST, Oden Institute for Computational Engineering and Sciences, The University of Texas at Austin — In LES, the unresolved turbulent stress is split into a mean and fluctuating portion, each with their own model. The AMS has shown improved accuracy in keeping computational cost low. Nevertheless, their predictive accuracy is limited by shortcomings such as modeled-stress-depletion and scalar grid measures. Haering, Oliver, and Moser developed an “active model-split” (AMS) to address many of these shortcomings. This model splits the unresolved turbulent stress into a mean and fluctuating portion, each with their own model. The AMS has shown improved accuracy in several incompressible test cases. In order to allow more complex test cases, this new model has been applied to the compressible flow equations and implemented in SU2. Results are presented for several compressible test cases, including supersonic channel and boundary layer flows and the Bachlo and Johnson transonic, axisymmetric bump.

5:32PM B19.00005 Hyperviscosity and bottlenecks in the Taylor-Green vortex, RAHUL AGRAWAL, IIT-Bombay, ALEXANDROS ALEXAKIS, MARC BRACHET, LPS-ENS, LAURETTE TUCKERMAN, PMNH-CNRS-ESPCI-Paris — Direct numerical simulations of turbulence are sometimes performed with hyperviscosity, in which the standard viscous term is replaced by a higher power \( p \) of the Laplacian, in order to increase the dissipation at high wavenumbers and thus to widen the computationally accessible inertial range. It is essential to determine the effect of hyperviscosity on various features of the turbulent energy spectrum, such as the bottleneck, an increase in energy for wavenumbers just below the dissipation range. Here, we use the symmetries of decaying Taylor-Green flow to study the effect of hyperviscosity on the bottleneck in high-resolution direct numerical simulations for resolutions up to \( 1024^3 \) and for hyperviscosity up to order \( p = 100 \), using simulations with \( 2048^3 \) and \( p = 1 \) as a reference case. We also investigate numerical issues that must be addressed for these high parameter values, in particular the timestepping scheme, the timestep, and the evaluation of the dissipation.

1This work was funded by the US Department of Energy's National Nuclear Security Administration under the Predictive Science Academic Alliance Program II at Stanford University, Grant-DE-NA-0002373.
We investigate an anisotropy-resolving subgrid-scale (SGS) model for large eddy simulation, which is constructed by combining an isotropic linear eddy-viscosity model (EVM) with an extra anisotropic term (EAT) (Abe, Int. J. Heat Fluid Flow, 39, pp. 42-52 (2013)). In this study, to reveal the role of the EAT in the SGS model, we performed simulations using several combinations for the terms involved in the SGS model, e.g., only EVM, only EAT as well as full version (EVM+EAT). We calculate the power spectrum of the GS velocity component from the obtained data and then compared them in detail. The comparison shows that the EAT works well for enhancing small-scale structures, resulting in an apparent upshift of the power spectrum particularly in a high-wavenumber region. We further investigate another option for the EAT instead of the currently-used formulation, e.g., a modified Leonard-stress formulation.

The present computer was primarily carried out using the computer facilities at the Research Institute for Information Technology, Kyushu University, Japan. This research was supported by JSPS KAKENHI Grant Number JP19K12005.

Asynchronous Direct Numerical Simulations (DNS) of turbulent flows at extreme scales. KOMAL KUMARI, DIEGO DONZIS, Texas A&M University — A major challenge in turbulence simulations is to accurately resolve the wide range of spatio-temporal scales. The computational cost of well resolved DNS grows with Reynolds number steeply and therefore, necessitates the use of massively parallel computations on supercomputers. However, the increase in communication and synchronization cost of current approaches could pose an insurmountable bottleneck at extreme scales. Thus, we have developed a novel paradigm, which relaxes these synchronization requirements at a mathematical level and leads to the so-called Asynchrony-Tolerant (AT) schemes. A first of its kind implementation of these schemes in a 3-D compressible Navier-Stokes DNS solver (forced and decaying) will be presented. Implementation of asynchrony using communication and synchronization avoiding algorithms resulting in periodic and random delays will be discussed. We show that these asynchronous algorithms accurately resolve large and small scale characteristics of turbulence, including instantaneous fields. We also show their efficiency in mitigating the communication bottleneck. The improved scaling, without affecting the physics, makes this asynchronous paradigm a path towards exascale simulations of turbulence and other non-linear phenomena.

A laboratory study of gas giants' zonal jets: formation and long-term evolution. DAPHNE LEMASQUERIER, BENJAMIN FAVIER, MICHAEL LE BARS, Aix-Marseille Univ, CNRS, Centrale Marseille, I.R.P.H.E, UMR 7342, 49 rue F. Joliot Curie, 13013 Marseille — The stability, structure and dynamics of gas giants' zonal jet streams are still poorly understood, especially in terms of coupling with their deep molecular interior. Here, we use an experimental approach to address the questions of zonal jets formation and long-term evolution. A strong topographic β-effect is obtained inside a rotating water tank thanks to the paraboloidal shape of the fluid free surface due the centrifugally-induced pressure. The shape of the bottom of the tank is chosen so that this β-effect is spatially constant. A small-scale turbulent forcing is performed over 128 injection and suction points at the base of the tank allowing a control of its spatio-temporal distribution and intensity. Time-resolving PIV measurements are performed in a horizontal plane. We identify a subcritical bifurcation between two regimes where zonal jets with strong instantaneous signature are present. In the first regime, the jets are weak in amplitude, directly forced and steady. In the second one, we observe self-developed highly energetic jets, independent of the forcing scale. In that last case, the zonal flows kinetic energy spectra are consistent with theoretical predictions in the regime of so-called zonostrophic turbulence relevant to planetary applications.

Shelf flow crossing over a strait: Experimental. JOSEPH KUEHL, University of Delaware, VITALII SHEREMET, NOAA — Motivated by the phenomenon of Scotian Shelf Crossover events (Bisagni et al. 1998), a problem of a shelf flow that is interrupted by a strait is considered. Laboratory experiments in a rotating tank with barotropic and baroclinic flow over flat and sloping shelves confirms that the flow is steered by the bathymetric contours and mainly circumnavigates the gulf. In order to jump across the strait, as suggested by earlier theories, the flow must have unrealistically high Rossby numbers. However, the near bottom friction relaxes the bathymetric constraint (Kuehl 2014) and causes the formation of a peculiar jet crossing the strait diagonally. Numerical solutions for realistic values of the frictional parameter reproduce the results of the laboratory experiments and are similar to the patterns observed in the satellite derived sea surface temperature fields (Smith 1983). Bisagni, J. J., R. C. Beardsley, C. M. Rusham, J. P. Manning, and W. Williams, 1996. Deep-Sea Research, 43, 1439-1471. Kuehl, J. J., 2014. Geophysical Research Letters, 41. Smith, P. C., 1983. Journal of Physical Oceanography, 13, 1034-1054.

Direct Statistical Simulations of a Rotating Thermal Annulus. JEF-FREY OISHI, Bates College, STEVEN TOBIAS, University of Leeds, BRAD MARSTON, Brown University, KEATON BURNS, Massachusetts Institute of Technology — Turbulent systems generate and interact with mean flows in a wide variety of natural systems. Understanding the nature of these interactions, particularly for systems far from equilibrium, remains a paramount priority in understanding large-scale flows on planets and stars. The fundamental problem in studying such systems via direct numerical simulation (DNS) is the fact that the smallest scales can have a significant impact on the mean flows, even when they are very widely separated. One way to make progress is to study the statistics of the flow rather than detailed flow variables themselves. By expanding around the mean flow in terms of equal-time cumulants, we can arrive at a closed set of equations of motion for the cumulants. Here, we present results using an expansion terminated at the second cumulant (CE2) for rapidly rotating thermal convection in an annulus. CE2 discards eddy-eddy interactions that yield eddies; it is fundamentally quasi-linear. We focus on a particular case in which the direct numerical simulation yields an initial three-jet solution that is unstable to a two-jet solution. Interestingly, CE2 predicts a stable three-jet solution, though we find that by biasing the initial conditions to favor certain symmetries, CE2 reproduces the DNS results.
5:32PM B20.00005 Mixing in Steady-State Gravity Currents, JIM MCELWAIN, Durham University; CLAUDIA CENEDESE, Woods Hole Oceanographic Institution, JEFFREY HENINGER, U. Texas at Austin — Turbulence currents have been observed to propagate for very long distances, longer than one would expect based on the current knowledge of mixing and evolution of gravity currents. Recent DNS simulations suggest that when in steady state the gravity current presents a much more stable interface, potentially reducing the mixing with ambient waters and hence being able to survive and propagate for longer distances. We report experiments that investigate experimentally 'steady state' gravity currents as opposed to lock-release gravity currents, with particular emphasis on the interfacial instability and assess whether the 'flavor' characteristics of this shear instability and the induced turbulence are different in steady state scenario as opposed to a more 'transient' scenario which has been investigated using lock-release gravity currents. We report on gravity current experiments in a flume that have reached a statistically steady state and compare the results to those obtained with a classic lock release set up, DNS and theoretical results.

5:45PM B20.00006 Effect of Inertial Migration of Particles on Flow Transitions of a Suspension in Taylor–Couette Geometry, LINA BAROUDI, ALEXANDER LIGAY, Manhattan College, MADHU MAJJI, University of California, Santa Barbara, JEFFREY MORRIS, CUNY City College of New York — This study presents an experimental investigation into the influence of inertial migration of neutrally buoyant non-Brownian particles on inertial flow transitions of a suspension in Taylor–Couette geometry. A concentric cylinder Taylor-Couette device with a stationary outer cylinder and rotating inner cylinder is considered. The device has an inner to outer radius ratio of 0.1. Starting with the circular Couette flow (CCF) regime with inertially migrated concentration profile in this regime, a lower onset Reynolds number observed when starting with uniform suspension concentration. For inertially migrated concentration profile in CCF, the non-axisymmetric flow states between CCF and Taylor vortex flow (TVF) reported by Majji et al. (JFM 835, 936 (2018)) were also observed here but at lower compared to uniform suspension concentrations. On contrary to the stabilizing effect obtained from the linear stability analysis of CCF with varying viscosity profile in the annular region, our results reveal a destabilizing effect to the CCF due to the inertially migrated concentration profiles.

5:58PM B20.00007 Vorticity shedding from a sphere moving vertically in a stratified fluid, HIDESHII HANAZAKI, TATSUYA YASUDA, KOKI TAKAGI, SHINYA OKINO, Kyoto University — A numerical study is described of the flow past a sphere moving at constant speeds in a stratified fluid. The wake under moderate or strong stratification remains axisymmetric even for $Re > 200$, unlike the flow of a homogeneous fluid which becomes asymmetric for $Re > 200$. A striking feature is that this axisymmetric flow often gives, under moderate stratification, an axisymmetric 'vorticity' shedding, which has never been observed in the wake of an axisymmetric bluff body. In this study, corresponding experiments have also been performed, showing that a non-axisymmetric wake with horizontal fluctuations tend to appear at some heights above the sphere. This suggests that the axisymmetric flow would be hard to be realised in actual flows at high Reynolds numbers ($Re > 200$).

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B21 Drops: Temperature and Marangoni Effects II 603 - Kirti Sahu, Indian Institute of Technology Hyderabad India

4:40PM B21.00001 Drop impact on heated nanostructures, LIHUI LIU, University of Alberta; Beihang University, GUOBIAO CAI, Beihang University, PEICHUN AMY TSAI, University of Alberta — Drop impact on a heated surface not only displays fascinating dynamics but also plays a crucial role in many industrial processes, such as spray cooling, metallurgy, and fuel injection. We experimentally investigate the impact dynamics of water droplets on both polished and nanostructured heated silica surfaces, under a wide range of surface temperature and impact speed (i.e., Weber number). For the polished surface, at a low surface temperature (below 200°C) water drop gently spread on the surface. Whereas, at a higher temperature (above the Leidenfrost point) the droplets completely rebound, with accompanying boiling, atomizing, and eventually splash as the Weber number increases. We show that the nano-structures significantly affect the impact dynamics, compared to that on the polished surface. In particular, the nano-textures can easily prompt splashing, vapor bursting, and jetting. We also found that the surface roughness and packing fraction of the nano-structures influence the impact outcomes.

1 NSERC Discovery and Canada Research Chair Program

4:53PM B21.00002 Marangoni spreading and contracting three-component droplets on completely wetting surfaces, NATE CIRA, DIETER BAUMGARTNER, SHAYANDEV SINHA, Rowland Institute at Harvard University, STEFAN KARPITSCHKA, Max-Planck Institute for Dynamics and Self-organization — Marangoni flows are a well-established mechanism for inducing droplet spreading and contraction. In this work, we study the behavior of a three-component mixture (ethanol, water, and propylene glycol) on high energy surfaces. Evaporation driven concentration differences give rise to surface tension gradients and Marangoni flows responsible for three categories of behavior across the ternary concentration space – contraction, enhanced spreading, and sequential spreading then contraction. Based on the combined effects of each component’s evaporation on surface tension, we predict boundaries for each of these behaviors that align well with experimental data across a wide range of concentrations and humidities. We present additional data on how internal flow structures impact droplet shape. Self-expansion and contraction make these droplets suitable for cleaning high energy surfaces.

1 Rowland Institute at Harvard University

5:06PM B21.00003 Solutal Marangoni Flows in Premixed Alcohol-water Solution, HYOUNGSOO KIM, KAIST, HAN SEO KO, Sungkyunkwan University — To date, the origin of solutal Marangoni flows in a premixed alcohol-water solution has not been investigated although the solutal Marangoni effects are widely used for various applications including material migration, mixing, coating and cleaning. It is reported that there is a flow transition during the evaporation of the premixed alcohol-water solution. Not from the pre-mixed alcohol-water mixture, we also show that the multiple irregular vortices in an evaporating water droplet were created where the cloud of alcohol molecules in the air covered the droplet, which induces a local surface tension variation. Interestingly, the flow pattern is also very similar to the case of the premixed water-alcohol solution droplet. By comparing these two different cases, we tried to explain how the solutal Marangoni flow occurs in the alcohol-water mixture solution.

1 This work was supported by Basic Science Research Program through the National Research Foundation of Korea funded by the Ministry of Science (NRF-2018R1C1B6004190 and NRF-2019M1A7A1A02089979). We also thank that this paper is based on research which has been conducted as part of the KAIST-funded Global Singularity Research Program for 2019.
Bouncing behaviors of an oil droplet in a stratified liquid

ZELAI XU, JUNYUAN CHEN, HAORAN LIU, HANG DING, University of Science and Technology of China

A liquid desiccant is a hygroscopic aqueous solution widely used in dehumidification processes. In this study, the temperature gradient affects the wetted length of the drop. The wetted length obeys an exponential function with time, but it diverges at the later stage due to the increase in the drop deformation. On the other hand, in the stretching regime, the drop continues to spread about its initial location as it experiences a higher surface tension force on the substrate. In contrast, in the displacement regime, the drop spreads slightly and moves towards one side of the substrate. In this regime, the drop experiences a lower apparent contact angle, opposing the prediction from evaporation-driven stabilization alone. Paired data from visualizing internal flow patterns for various liquids and substrates show that these effects on angle are consistent with observed Marangoni flows. These results underscore the importance of considering Marangoni flow in understanding the shape of evaporating highly volatile droplets on fully wetting surfaces.

Marangoni flow influences apparent contact angle of single component volatile liquids on completely wetting surfaces

SHAYANDEV SINHA, Rowland Institute at Harvard, Harvard University

Dhuynevolatile single-component liquids on completely wetting surfaces, we observe two scaling laws, with the wetted length of the drop increasing with time, $t$. The present work provides information on the physics of the interaction of droplets with a free surface and presents a new methodology for modeling triple contact lines. This method can be used in a wide range of applications involving three immiscible liquids.

Marangoni flow influences apparent contact angle of single component volatile liquids on completely wetting surfaces

ZHENYING WANG, Kyushu University, GEORGE KARAPETSAS, Aristotle University of Thessaloniki, PRASHANT VALLURI, ADAM WILLIAMS, University of Edinburgh, KHELLIL SEFIANE, Kyushu University; University of Edinburgh, YANSHEN LI, CHRISTIAN DIDDENS, LIJUN THAYIL RAJU, University of Twente, XUEHUA ZHANG, University of Alberta, KAI LEONG CHONG, University of Twente, ANDREA PROSPERETTI, University of Houston, DETLEF LOHSE, University of Twente — As we had found in Li et al. (2019), an oil droplet is able to repeatedly first sink and then bounce up in a vertically stratified ethanol-water mixture. The Marangoni flow at the droplet-liquid interface, caused by the solute (ethanol) gradient, provides the propulsion for the upwards jump. We now further elucidate the mechanism and explore the phase space. Specifically, we find that as the droplet jumps up to the lighter ethanol-rich region, it gets harder for the propelling droplet to overcome its own weight, until finally it stops. At this maximum height, the Marangoni flow continues to propel, homogenizing its surrounding liquid, thus decreasing the propulsion itself, until it ceases. The droplet then sinks while dragging down this uniform liquid layer with it. We find that this uniform “shielding” layer eventually vanishes, mainly because of diffusion, so that the droplet bounces up again, continuing the cycle. It is also found that the maximum height increases with decreasing the droplet size.

We thank the Rowland Fellow program for funding this work.

Motion of self-rewetting drop on a substrate with a linear temperature gradient

ZELAI XU, JUNYUAN CHEN, HAORAN LIU, HANG DING, University of Science and Technology of China

The present work provides information on the physics of the interaction of droplets with a free surface and presents a new methodology for modeling triple contact lines. This method can be used in a wide range of applications involving three immiscible liquids.

Lubrication model for vapor absorption into hygroscopic liquid desiccant droplets

ZHENYING WANG, Kyushu University, GEORGE KARAPETSAS, Aristotle University of Thessaloniki, PRASHANT VALLURI, ADAM WILLIAMS, University of Edinburgh, KHELLIL SEFIANE, Kyushu University; University of Edinburgh, YANSHEN LI, CHRISTIAN DIDDENS, LIJUN THAYIL RAJU, University of Twente, XUEHUA ZHANG, University of Alberta, KAI LEONG CHONG, University of Twente, ANDREA PROSPERETTI, University of Houston, DETLEF LOHSE, University of Twente — As we had found in Li et al. (2019), an oil droplet is able to repeatedly first sink and then bounce up in a vertically stratified ethanol-water mixture. The Marangoni flow at the droplet-liquid interface, caused by the solute (ethanol) gradient, provides the propulsion for the upwards jump. We now further elucidate the mechanism and explore the phase space. Specifically, we find that as the droplet jumps up to the lighter ethanol-rich region, it gets harder for the propelling droplet to overcome its own weight, until finally it stops. At this maximum height, the Marangoni flow continues to propel, homogenizing its surrounding liquid, thus decreasing the propulsion itself, until it ceases. The droplet then sinks while dragging down this uniform liquid layer with it. We find that this uniform “shielding” layer eventually vanishes, mainly because of diffusion, so that the droplet bounces up again, continuing the cycle. It is also found that the maximum height increases with decreasing the droplet size.

We thank the Rowland Fellow program for funding this work.
4:53PM B22.00002 Partial coalescence of a droplet with a pool of different viscosity\(^1\).

ABDULLAH AL ALHARETH, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology — The partial coalescence of a droplet with a liquid surface is a critical factor during the coarsening of emulsions, as this process can produce smaller droplets. It is well-known that increased liquid viscosity or reduced drop size will eventually stop the satellite pinch-off at a critical Ohnesorge number. On the other hand, similar satellites can be pinched off during the rapid spreading of a drop on a solid surface, if it is strongly hydrophilic. Herein we study this process when the substrate transitions from low to high viscosity liquid, modeled to approach the boundary conditions of a solid. We use miscible silicone oils, with a low-viscosity drop and a large range of pool viscosities up to a million times that of the drop. We observe non-monotonic behavior and as the viscosity increases we see a propensity for second-stage pinch-off. We connect the observed critical Ohnesorge numbers to the spreading behavior. 1. Zhang, Li & Thoroddsen, Phys. Rev. Lett. 102,104502 (2009).

\(^1\)This work was funded by King Abdullah University of Science and Technology (KAUST) under Grant No. URF/1/2621-01-01.


SWATI SINGH, ARUN K. SAHA, Indian Institute of Technology Kanpur, Kanpur, India — The drop coalescence is important in many applications such as formation of rain drops in cloud and mixing in microdevices. In the present work, the dynamics of satellite drop generation during the coalescence of two drops of unequal size are studied. The mechanism depends on five non-dimensional parameters: Ohnesorge number for both liquids, Bond number, Atwood number and the diameter ratio of two drops. We have performed two-dimensional, axisymmetric simulations using Coupled Level set and Volume of fluid method (CLSVOF) to unveil the underlying physics of coalescence process under varying drop diameter ratio (1.0-9.0) and Ohnesorge number. The Bond number is kept less than 0.1. Result shows the three different pinch-off scenario depending on drop diameter ratio: (i) the mother drop deforms after coalescence resulting in necking and subsequent satellite drop pinches off following the similar sequence as that of a drop coalescence with a flat liquid pool, (ii) occurrence of mother drop pinch-off happens with necking but without intermediate detachment of satellite drop, (iii) mother drop completely coalesces into the father drop without any evidence of satellite drop generation.

5:19PM B22.00004 Two Fluid Coalescence of Drops in an Exterior Fluid.

CHRISTOPHER ANTHONY, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — The coalescence of drops and bubbles has been investigated in numerous experimental, numerical and theoretical studies due to its prevalence in industrial and natural processes. In many studies to date, either the interior/dispersed phase (e.g. the gas inside bubbles) or the exterior/continuous phase (e.g. the air surrounding drops) has been treated as dynamically inert in order to simplify the analysis. However, in many systems of interest, the interior and the exterior phases are fluids of comparable viscosities and/or densities, and treating one as dynamically inert may be inappropriate. Indeed, theoretical arguments by Eggers et al. suggest that the interior fluid will always play a role at some early time but that this could occur at length scales below the continuum limit (e.g. with bubbles). To date, the scaling behavior predicted by Eggers et al. for viscous drops in a viscous medium has been contradicted by experimental observations and no numerical study has been able to resolve this contradiction. Here, we explore through high fidelity numerical simulations drop coalescence in an exterior medium paying particular attention to the high viscosity or creeping flow limit while also considering a limited number of cases of finite viscosity.

5:32PM B22.00005 Non-continuum tangential resistivity functions for two spheres suspended in a gas\(^1\), MELANIE LI SING HOW, DONALD L. KOCH, LANCE R. COLLINS, Cornell University — In the limit of absence of gas-phase inertia, the motion of two spheres is described by the Stokes equation for large separations and by the linearized Boltzmann equation for separations comparable with the mean-free path of the gas. The forces and torques the gas exerts on each sphere are described by five resistivity functions (Jeffrey & Onishi, J. Fluid Mech. 139: 261-290 1984). The resistivity functions undergo a transition as the separation distance between the spheres approaches the mean free path of the gas. In particular, the divergences of the Stokes resistivity functions as the spheres approach contact are reduced by non-continuum effects. We present modified tangential resistivity functions that are uniformly valid through this transition (i.e., from the continuum limit at distant separations to separations where the lubrication flow near contact is a free molecular flow). We apply the modified resistivity functions to two illuminating cases: (i) a sphere falling in close proximity to a vertical wall; and (ii) two spheres settling under gravity. The results show qualitative differences from the classical Stokes flow solution and are applied to the study of coalescing cloud droplets settling under gravity.

\(^1\)CBET - 1605195

5:45PM B22.00006 Coalescence dynamics of two unequal size water droplets in air.

NITIN GOYAL, ATUL SHARMA, Indian Institute of Technology Bombay — Coalescence dynamics of two water droplets falling under gravity in air, is studied for various diameter ratio \(d_{\text{bottom}}/d_{\text{top}}\). \(D_r = 0.156-0.4\) and Ohnesorge number \(Oh = 0.001-0.025\) at a Bond number \(Bo = 0.092\). Axisymmetric simulations are done using sharp interface level set method based in-house code. We presented a regime map for the various \(Oh\) and \(D_r\), with partial coalescence at smaller as well as larger values of \(D_r\) and full coalescence at intermediate value of \(D_r\). For a transition in the coalescence dynamics, the present work reports a smaller and a larger critical value of \(D_r\) as compared to one critical \(D_r\) (the larger one) reported in the literature. Over the range of \(Oh\) studied here, there is a monotonic increase in both the critical \(D_r\) with increasing \(Oh\) from 1.5 to 5.7 for the larger critical \(D_r\) and from 0.3 to 0.4 for the smaller critical \(D_r\). The pinch-off height of satellite droplet is larger for partial coalescence at the larger \(D_r\) as compared to that at the smaller \(D_r\). The partial coalescence regime at the smaller \(D_r\) is presented here for the first time.

Saturday, November 23, 2019 4:40PM - 6:11PM — Session B23: Drops: Electric Field Effects I

4:40PM B23.00001 Electrohydrodynamic Equatorial Streaming

BRAYDEN WAGONER, CHRISTOPHER ANTHONY, Purdue University, PETIA VLAVHOVSKA, Northwestern University, MICHAEL HARRIS, OSMAN BASARAN, Purdue University, SCHOOL OF CHEMICAL ENGINEERING, PURDUE UNIVERSITY TEAM, ENGINEERING SCIENCES AND APPLIED MATHEMATICS, NORTHWESTERN UNIVERSITY TEAM — When subjected to electric fields, spherical liquid drops can deform, and disintegrate by fissioning, cone-jetting, and in a variety of other ways. The strength of the electric field as well as the electrical and other physical properties of the drop and surrounding medium determine both the extent and type of deformation (prolate/oblate) that the drops can undergo prior to disintegration. At large electric field strengths, prolate drops emit thin jets from conical structures (Taylor cones) that form at their poles. Oblate drops, on the other hand, may burst at their centers (dimpling) or emit a thin sheet from their equators (equatorial-streaming) [Brossoue and Vlahovska, Phys. Rev. Lett., vol. 119, 2017]. We probe the physics behind these two oblate instabilities through direct numerical simulation.
Numerical simulations on the extended lubrication theory is developed in order to predict the bubble velocity and steady state solutions of the film profiles in different directions. By varying the capillary number of the external downward flow, different film profile regimes are observed. A theoretical model based on electrowetting can be used as a macroscopic probe into the structure of the EDL. Not all drops of commercial hydrogel, agar-agar, alginate, xanthan gum, and gum Arabic can be moved by electrowetting. Drops of sizes of 200 \textmu{}m and several aqueous polymer solutions, like those of polyethylene oxide and polyacrylamide result in drop motions similar to that of water drops. Several aqueous polymer solutions, like those of polyethylene oxide and polyacrylamide result in drop motions similar to that of water drops. The spreading time to reach this final state varies as \( c_0^{-1} \) and the receding time varies as \( c_0^{-1/2} \). We show that both spreading and receding times are governed by the formation of the electrical double layer (EDL) on the substrate, and that electrowetting can be used as a macroscopic probe into the structure of the EDL.

Ambient Air Humidity Affects the Charge Acquired by Water Drops in Oil. By varying the capillary number of the external downward flow, different film profile regimes are observed. A theoretical model based on electrowetting can be used as a macroscopic probe into the structure of the EDL.

Dynamics of electrowetting-driven spreading and receding of a droplet. By studying the effect of electrolyte concentration on electrowetting-induced spreading and receding of a droplet on the basal plane of graphite, a smooth conducting substrate. We find that for positive applied potentials, the static electrowetting contact angle contact angle is independent of \( c_0 \) over three decades of concentration but that the spreading time to reach this final state varies as \( c_0^{-1} \). In contrast, for negative potentials, the electrowetting angle varies as \( 1/c_0 \) and the spreading time varies as \( c_0^{-1/2} \). We show that both spreading and receding times are governed by the formation of the electrical double layer (EDL) on the substrate, and that electrowetting can be used as a macroscopic probe into the structure of the EDL.

Drop Manipulation by Electrowetting for 3D Printing. Drops of commercial hydrogel, agar-agar, alginate, xanthan gum, and gum Arabic can be moved by electrowetting. Drops of sizes of 200 \textmu{}m and 3 mm can be manipulated by the electric field on different dielectric substrates accurately and repeatedly. This includes horizontal motions, motions on vertical wall, and upside down. Theoretical aspects are discussed as well.

Rising or Sinking — An Elongated Bubble in a Vertical Capillary Tube Under External Flow. When a confined bubble translates steadily in a vertical capillary tube under co-current fluid flow, the motion of the bubble depends on both the buoyancy and the mean fluid velocity. Although a wide variety of studies have been carried out analyzing the motion of the elongated confined bubble, most of them focus on the dynamics in one of two distinct limit — either when buoyancy effects are negligible (\( Bo << 1 \)) or when \( Bo >> 1 \). This work, we systematically investigate the motion of an elongated bubble and its film thickness profile with \( Bo = 1 \) under external downward flow. In a stagnant fluid, a bubble spontaneously rises only when \( Bo > 0.842 \) (Bretherton 1961), and intuitively, external downward flow can slow down the bubble motion or even reverse the translation direction. By varying the capillary number of the external downward flow, different film profile regimes are observed. A theoretical model based on the extended lubrication theory is developed in order to predict the bubble velocity and steady state solutions of the film profiles in different regimes. The critical external flow conditions are further characterized and validated by combining with the results from experiments and direct numerical simulations.

---

1 National Science Foundation (NSF) Grant 1825626.

---

We thank the NSF for support via grant CBET-1804863. YEF thanks the PEI for support via Mary and Randall Hack 69 Research Fund.
4:53PM B24.00002 Effect of wake behind a rising bubble on the heat transfer of the vertical heated wall\(^1\)  Hwiyoung Maeng, Hyungmin Park, Seoul National University — We conduct experiments with two kind of bubbles (linear and zigzagging bubble of which Reynolds number is 200 and 850) rising near the vertical heated (at a constant heat flux of 2000 W/m\(^2\)) wall. While varying the distance between the bubble and the wall, we measure the gas- and liquid-phase flow fields with two-phase PIV and wall temperature distribution using an infrared camera simultaneously. We find out that the wake emanating behind the zigzag bubble convects laterally toward the wall, disturbing the natural convective boundary layer and enhances the local convective heat transfer, which is quantified as a Nusselt number (Nu) normalized by that of natural convection. As the distance between the bubble and the wall is smaller, the increasing slope and affected area of the heat transfer become larger. The maximum Nu augmentation is eight times larger than the reference value when the bubble-wall distance is short enough for the bubble to collide with the wall, but the affected area is maximum when the bubble does not collide but passes closely to the wall. In the case of linearly rising bubble, however, the heat transfer enhancement was not measured evidently, because the wake is stationary and does not convect to the vertical wall effectively.

\(^1\)Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

5:06PM B24.00003 Numerical study on the interaction of two bubbles rising in a power-law fluid  Shu Takagi, Varun Jadon, The University of Tokyo, Kazuyasu Sugiyama, Osaka University — In this study, interactions of two bubble rising in a Non-Newtonian power-law fluid have been investigated through the direct numerical simulations. An interface capturing method with a continuous function is introduced and the three-dimensional model based on modified VOF method is used to study two-bubble interaction phenomena in a power-law fluid. Here, we discuss the influence of shear-thinning and shear-thickening characteristics on the interaction of in-line or inclined configurations of two bubbles. The numerical results reveal that the shear-thinning fluid enhances the inviscid potential interaction and in-line motion becomes more unstable in shear-thinning fluid than that in Newtonian Fluid.

5:19PM B24.00004 Effect of viscosity and density ratios on two bubbles rising side-by-side  Sivanandan Kavuri, Mounika Balla, Indian Institute of Technology Hyderabad, India, Manoj Kumar Tripathi, Indian Institute of Science Education and Research Bhopal, Kirti Sahu, Indian Institute of Technology Hyderabad, India, Rama Govindarajan, International Centre for Theoretical Sciences, Bangalore — We study the dynamics of a pair of initially spherical ‘bubbles’ of fluid rising side-by-side in a surrounding, denser, fluid. Interesting dynamics are reported, which cannot be extrapolated from previously known dynamics of gas-liquid systems. Similar to two air bubbles though, we find that two liquid bubbles move away from each other as they rise, in cases where a single bubble would rise vertically. A pair of light bubbles always remains in two-dimensional motion, and higher bubble viscosity increases the tendency of wobbling. This is in contrast with the dynamics of a single bubble that follows a highly three-dimensional trajectory at very low bubble viscosity, but is restricted to two dimensional motion at higher bubble viscosity. On the other hand, a pair of heavier bubbles displays three-dimensional behaviour at low bubble viscosity and two dimensional behaviour at high viscosity. We find that a pair of bubbles is far less sensitive to viscosity contrast than a single bubble is, in our parameter range. In contrast to gas-liquid systems, where shape change of the bubble was tied to nonlinear dynamics of the trajectory, we find in liquid-liquid systems that interesting bubble trajectories can occur without corresponding large shape changes.

5:32PM B24.00005 On the Dynamics of an Oil Encapsulated Bubble  Joel Karp, Ernesto Mancilla, Rigoberito Morales, Federal University of Technology-Parana — The dynamics of an oil encapsulated bubble was experimentally investigated. High-speed imaging was employed in the visualization of the motion of a coated bubble rising in quiescent tap water. The attachment of a bubble with diameter ranging from 300 to 3000 m to a 2.3 mm oil droplet resulted in two different regimes: oscillatory and non-oscillatory motion. The influence of the properties of the oil coating was investigated by employing three different oils, which were found to be more influential when no motion instabilities were observed and viscous forces dominate. Path oscillations decreased for the encapsulated bubble, being up to five times smaller in comparison with an isolated bubble with similar diameter. The oil coated bubble presented higher terminal velocity in comparison to a rising oil droplet, with increase factors from 100 up to 400%. The shape deformation of the encapsulated bubble was less than 15% within the diameter range evaluated, being essentially spherical in comparison to corresponding isolated bubbles. Transient motion assays indicated that developed regime was achieved in approximately half the rising distance than for isolated bubbles, after which a periodic oscillation of the velocity components was observed.

5:45PM B24.00006 Flow boiling in a Horizontal Microchannel with high Aspect Ratio for Non-Uniform (One-Sided) Heating at Different Rotational Orientations\(^1\)  Marius Vermaak, Jaco Dirker, University of Pretoria, Khellil Sefiane, Daniel Orejon, Prashant Valluri, University of Edinburgh, Josua Meyer, University of Pretoria, UNIVERSITY OF PRETORIA TEAM, UNIVERSITY OF EDINBURGH COLLABORATION — Flow boiling of FC72 in a semi-rectangular horizontal microchannel with one-sided heating, was investigated experimentally. Microchannel rotational orientations from bottom-heating (0) to top-heating (180), in 30\(^\circ\) increments were considered. Mass fluxes of 10, 20 and 40 kg/m\(^2\)s, paired with various heat fluxes which spanned from the onset of nucleate boiling to near dry-out conditions, were tested in a borosilicate glass channel of 5x0.5mm internal cross-section. Ohmic resistive heating was obtained via a layer of tantalum oxide that was applied to only one of the outer surfaces of the microchannel. This optically transparent layer allowed for both transient surface temperature mapping and flow visualization. During quasi steady state average and local heat transfer coefficients as well as pressure drop were analyzed. Our results show that bottom-heating produced average heat transfer coefficients at least 17% higher than top-heating. Flow boiling instabilities are promoted at greater wall super-heats, especially in rotations between top-heating and side-heating. The effect of inclination on flow boiling in such microchannels will also be presented.

\(^1\)ThermaSMART Project
We acknowledge financial support from the Engineering and Physical Sciences Research Council of the UK via grant No. EP/R041954/1.
and BELSPO for funding. PC gratefully acknowledges the support of the F.R.S.-FNRS.

1 adsorption/absorption) of water in the drop. It has been observed experimentally that in these cases, the ambient relative humidity (RH) and the initial drop composition seems to play a very important role in the wetting dynamics. On varying the composition of IPA/water mixture and alcohol [e.g., Iso-Propanol (IPA)] and water. It has been observed experimentally that in these cases, the ambient relative humidity (RH) and the initial drop composition seems to play a very important role in the wetting dynamics. On varying the composition of IPA/water mixture and RH, with the help of optical interferometry, the droplet spreading behavior has been studied. For better understanding, an analytical model has been developed to characterize the dependence of these parameters on wetting behavior. The model has been conveniently used to generate so-called a map of regime (MOR) diagram which clearly demarcates the different wetting zones depending on the relative evaporation (or adsorption/absorption) of water in the drop.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B26 Drag Reduction I

4:40PM B26.00001 Active control of deterministic turbulent spots for drag reduction1, KWING-SO CHOI, YAXING WANG, University of Nottingham, MICHAEL GASTER, CHRIS ATKIN, City, University of London, YURY KACHANOY, VLADIMIR BORODULIN, ITAM — A series of experiments was carried out using Gaster’s wind tunnel at City, University of London, where the freestream turbulence level in the test section was extremely low (0.006% between 2 Hz and 2 kHz). With weak excitations applied from spanwise-periodic 19 miniature phasers located downstream of a flat-plate leading edge, the boundary-layer development was studied in detail using a hot-wire anemometer at laminar, transitional and turbulent stages. Careful velocity measurements revealed an appearance of turbulent spots, which were precisely reproducible in both time and space each time the pseudo-random signal was applied. The emergence of turbulent spots, which bypassed a full development of T-S waves, was deterministic at least in the lower frequency range of velocity signals, enabling us to examine their structure that was not possible before. Opposition control was then carried out by issuing wall-normal jet on the high-speed region of turbulent spots with a view to achieve a skin-friction reduction by delaying transition to fully-developed turbulence. This was done without sensors as all boundary-layer structures were “deterministic”.

4:53PM B26.00002 ABSTRACT WITHDRAWN –

5:06PM B26.00003 Active Drag Reduction in Turbulent Airfoil Flow. WOLFGANG SCHRDER, MARIAN ALBERS, Institute of Aerodynamics, RWTH Aachen University — Drag reduction in turbulent boundary layers is key for substantial energy savings in aerodynamics. Large parts of the flow over the wing of modern aircraft are turbulent such that even net energy savings of a few percent lead to high cost savings. Active drag reduction methods have shown to be capable of significantly reducing the drug in generic external turbulent wall-bounded flows. Based on the knowledge of previous studies for flat-plate turbulent boundary layer flow the technique of spanwise traveling transversal surface waves is applied to 74 percent of the surface of a NACA4412 wing section at a chord-based Reynolds number of $Re_c = 400,000$. Different parameter combinations are tested for maximum drag reduction and maximum net power saving. The results show a reduction of the total drag of up to 8.5 percent and a decrease of the viscous drag by up to 12.9 percent. Note that this includes all actuated and non-actuated parts of the surface, i.e., locally a much higher decrease of the wall-shear stress is achieved. Additionally the lift is slightly increased by up to 1.4 percent.

5:19PM B26.00004 Active flow control of the logarithmic layer1, ANNA GUSEVA, MIGUEL P. EMUNAR, JAVIER JIMENEZ. School of Aeronautics, Polytechnic University of Madrid — Active flow control for years has been a vivid topic of fluid dynamics research. It is of especial importance for wall-bounded turbulent flows, where intense dissipation at the wall can produce undesirable effects. One successful control approach is to apply at the boundary a velocity field opposite to the observed in the buffer layer. The focus of this work is on creating a control strategy that can be reproduced in experimental facilities. By acting on the flow from the wall, we aim to affect the eddies of relatively large wavelengths ($h \approx 0.1$) at $Re_c = 1000$. We reconstruct the wall-normal velocity in the log-layer ($y^+ > 100$) with the linear stochastic estimation method. Preliminary implementation of opposition control on the large scales results in substantial drag increase, indicating that we are able to significantly affect those scales and have plenty of control authority. In the conference we present further development of this strategy. We compare the control efficiency of applying different wavelength bands at the wall and check the impact of imposed boundary conditions on the statistics of velocity fluctuations, as well as new structures created in the flow. Finally, we assess the general applicability of our control to the existing measurement techniques.

1 European Research Council, ERC-2014.AdG-669505

5:32PM B26.00005 Drag Reduction in the Flow around a Cylinder: A Bayesian Optimization Approach1, ANTHONY LARROQUE, MIGUEL FOSAS DE PANDO. Department of Mechanical Engineering and Industrial Design, University of Cadiz, LUIS LAFUENTE, Department of Mathematics, University of Cadiz — Bayesian Optimization has recently gained popularity as an effective tool to deal with expensive black-box objective functions. Advantages of this method include the ability to determine the global minimum at a reduced number of iterations, and the possibility to include uncertainty in the evaluation of the objective function. Recent developments also include parallel function evaluations or multiple sources of information with varying fidelity. All these features render Bayesian Optimization a promising tool for Direct Numerical Simulations or Large Eddy Simulations, where the computational cost of determining the cost function, such as the drag coefficient, is typically very high, and the computational resources are very limited. In this work, we consider the three dimensional flow around a cylinder and apply Bayesian Optimization to determine the optimal blowing and suction strategy at the wall that leads to a minimum drag coefficient. The efficiency of the resulting optimization scheme is compared to alternative methods. Finally, we discuss the optimal parameters of the velocity profile, the underlying physical mechanisms, and the influence of the Reynolds numbers on the optimal solutions.

1 The authors gratefully acknowledge financial support from MINECO through the Programa Estatal de I+D+i Orientada a los Retos de la Sociedad, Grant No. DPI2016-77777-R, MINECO AEI/FEDER, UE.
Examining the instantaneous and time-averaged velocity distributions, we discuss the relevance of the Stokes boundary layer near the sphere is used to study the physical mechanisms driving DR, which we report between 20% and 30% for $\lambda$. Wings may compensate for changes in spanwise vorticity that induce vortex bursting. Also disrupted on stiffened wings, suggesting that the loss of flexibility gradients contributes to LEV bursting. Flexibility gradients on insect rigid wings, the LEV often bursts but remains attached at Re with well-defined reattachment streaklines. After desiccating, the wings stiffen in both the chord- and spanwise direction, less than the local chord length with well-defined reattachment streaklines. Decreasing the viscosity of the lubricating layer ($\lambda<\lambda_0=4$) from a main layer of fluid (viscosity $\eta_1$) to a thin layer of fluid (viscosity $\eta_2$). To single out the effect of surface tension, we focus initially on two fluids having same density and viscosity, and we then consider a wide range of viscosities of the lubricating layer: from $\lambda=\eta_1/\eta_2=0.25$ (less viscous) up to $\lambda=\eta_1/\eta_2=4.00$ (more viscous). A database comprising DNS of two-phase flow channel turbulence is used to study the physical mechanisms driving DR, which we report between 20% and 30% for $\lambda \leq 1$, 10 % for $\lambda = 2.00$ and absent for $\lambda = 4.00$. The maximum DR occurs when the two fluids have the same viscosity ($\lambda = 1$), and corresponds to the laminarization of the lubricating layer. Decreasing the viscosity of the lubricating layer ($\lambda < 1$) induces a marginally decreased DR, but also helps sustaining strong turbulence in the lubricating layer. This led us to infer two different mechanisms for the two drag-reduced systems, each of which is ultimately controlled by the outcome of the competition between viscous, inertial and surface tension forces.

Saturday, November 23, 2019 4:40PM - 5:58PM — Session B27 Biological Fluid Dynamics: Insect Flight I - Wing Properties 609 - Kai Schneider, Aix-Marseille University

4:40PM B27.00001 Vortex bursting corresponds to reduced gradients in wing flexibility, MEGAN MATTHEWS, BRYAN MCCARTY, SIMON SPONBERG, Georgia Tech — The leading-edge vortex (LEV) is a well-known flight mechanism for flapping insects, but the interplay between the boundary LEV and the flexible wing it attaches to is not yet understood. On rigid wings, the LEV often bursts but remains attached at Re ~ O(10^4). However, this has not been seen on flexible insect wings. Force production increases with decreasing flexural stiffness and further increases when the wing exhibits chord- and spanwise gradients in flexural stiffness. We mounted real hawkmoth wings onto a motor that revolves the wings at a constant frequency and generates a coherent LEV. Using smoke-wire visualization, we observed the qualitative structure of the LEV. On freshly-mounted wings, LEV diameter is consistently 50% or less than the local chord length with well-defined reattachment streaklines. After desiccating, the wings stiffen in both the chord- and spanwise directions, and the LEV formed on stiffened wings has a diameter of at least 80% of the local chord length. The reattachment streaklines were also disrupted on stiffened wings, suggesting that the loss of flexibility gradients on insect wings may compensate for changes in spanwise vorticity that induce vortex bursting.

4:53PM B27.00002 ABSTRACT WITHDRAWN — 5:06PM B27.00003 Unsteady maneuvering of a morphing wing., KAMLESH JOSHI, SAMIK BHATTACHARYA, University of Central Florida — The unsteady flow over a morphing flat-plate airfoil is investigated in this work. The unsteady flow is generated by accelerating the wing from rest and from one steady speed to a higher speed. The plate can be bent smoothly along the span with a flexion ratio of 0.7 and a flexion angle of 20 degree. In these tests, two different bending rates (BR), namely BR = 1s and 2s were implemented. The wing was towed from rest to Reynolds numbers of 10,000 and 20,000, and it was bent simultaneously along the span with zero phase difference between the forward towing motion and the bending motion. Instantaneous forces were measured with a six-dof force sensor, and the flow field was measured with the help of particle image velocimetry. It was found that spanwise bending has a considerable effect on the unsteady forces during acceleration. The vortex dynamics of the leading-edge-vortex was altered due to the variation of the shear layer velocity along the span which occurred due to the bending motion. In this work, we sought to quantify the effect of bending rate on the stability of the leading-edge-vortex. We show that the gradual lifting of the tip vortex closer to the LEV, affects its growth. We also demonstrate that the spanwise bending action alters the added mass peak due to the inertial forces caused by the bending process.

5:19PM B27.00004 Effect of chordwise wing flexibility on the flapping flight of a butterfly-like 3D flapping wing-body model, KOSUKE SUZUKI, TAKAAKI AOKI, MASATO YOSHINO, Shinsyu University — In our recent study [K. Suzuki, T. Aoki, and M. Yoshino, Phys. Rev. E 100, 013104 (2019)], we constructed a flexible wing with chordwise flexibility by connecting two rigid plates with a torsion spring, and investigated the effect of chordwise wing flexibility on the flapping flight of a simple butterfly-like flapping wing-body model by using an immersed boundary-lattice Boltzmann method. First, we investigated the effects of the spring stiffness on the aerodynamic performance when the body of the model is fixed. We found that the time-averaged lift and thrust forces are larger than those of the rigid wings. Second, we simulated free flights when the body of the model can only move translationally. We found that the model with the flexible wings at an appropriate value of the spring stiffness can fly more effectively than the model with the rigid wings, which is consistent with the results when the body of the model is fixed.
5:32PM B27.00005 Numerical simulation of insect flight with flexible wings using a mass-spring fluid-interaction solver. KAI SCHNEIDER, HUNG TRUONG, Aix-Marseille Univ, THOMAS ENGELS, LMD-IPSL, Ecole Normale Superieure-PSL, DMITRY KOLOMENSKY, Japan Agency for Marine-Earth Sci and Tech (JAMSTEC), THE AIFIT TEAM — Fundamental characteristics of insect flight are flexible wings, which play an important role for their aerodynamics. Real wings are delicate structures, composed of veins and membranes, and can undergo significant deformation. Here we present detailed numerical simulations of such deformable wings modeled by a mass-spring network. The mass-spring model uses a functional approach, thus modeling the veins and the membranes of the wing. Results are obtained with a fluid-structure interaction solver, coupling a mass-spring model for the flexible wing with the pseudo-spectral code FLUSI solving the incompressible Navier-Stokes equations. We impose the no-slip boundary condition through the volume penalization method; the time-dependent complex geometry is then completely described by a mask function. We perform a series of numerical simulations of a flexible revolving bumblebee wing at a Reynolds number Re=1800. In order to assess the influence of wing flexibility on the aerodynamics, we vary the elasticity parameters and study rigid, flexible and highly flexible wing models. A better aerodynamic performance of the flexible wing, characterized by the increase of the lift-to-drag ratio, is found while the highly flexible wing appears to be less efficient than the rigid wing.

1Financial support from the ANR (Grant No. 15-CE40-0019) and DFG (Grant No. SE 824/26-1), project AIFIT, is gratefully acknowledged. The authors acknowledge HPC resources of IDRIS (No. 2018-01664) by GENCI. D.K. gratefully acknowledges financial support from the JSPS KAKENHI Grant No. JP18K13693.

5:45PM B27.00006 Effect of torsional stiffness on passive wing pitch and its aerodynamic performance in hovering flight. MENGLONG LEI, CHENGYU LI, Villanova University — Insect’s wings are able to passively maintain a high angle of attack due to the torsional flexibility of wing basal region without the aid of the active pitching motion. However, there is no clear understanding of how torsional wing flexibility should be designed to achieve optimal aerodynamic performance. In this work, a computational study was conducted to investigate the passive pitching mechanism of a fruit fly wing in hovering flight using a torsional spring model. The torsional wing flexibility was characterized by Cauchy number. Different flapping patterns including zero-derivation, figure-8, oval-shaped stroke kinematics were evaluated. The aerodynamic forces and associated unsteady flow structures were simulated using an in-house immersed-boundary-method based computational fluid dynamic solver. Our simulations revealed that the optimal lift and lift-to-power ratio can be achieved in a particular range of Cauchy number (0.16~0.30) regardless of its stroke kinematics. This range is consistent with the Cauchy number calculated based on the fruit fly data from literature. The findings of this work could provide important implications for designing more efficient flapping-wing micro air vehicles.

1This study is supported by 2019 Villanova University Summer Grant Program and 2019 ORAU Ralph E. Powe Junior Faculty Enhancement Award to CL.

Saturday, November 23, 2019 4:40PM - 6:11PM –
Session B28 Turbulence Theory: Spectral Transfer
601 - Stavros Tavoularis, University of Ottawa

4:40PM B28.00001 Fluctuations in spectral energy transfer and their consequences
SUALEH KHURSHID, DIEGO DONZIS, Department of Aerospace Engineering, Texas A&M University, KATEPALLI SREENIVASAN, New York University — The assumption in Kolmogorov’s seminal work, as well as in all the subsequent work of that genre, is that the average energy transfer across scales is unidirectional, from the large to the smaller, in the case of homogeneous turbulence, from low wavenumbers to high wavenumbers. This assumption would be quite acceptable if the fluctuations in the energy transfer rate were small compared to the mean. But the energy transfer rate is a highly fluctuating quantity, which is nearly as often in the backward direction as forward, and the mean transfer is only a small difference between the two, suggesting the need to consider fluctuations explicitly. In this work, we characterize the fluctuations in wavenumber space, of the energy itself and energy transfer rate, for a range of Reynolds numbers, using highly resolved direct numerical simulations. The simulations allow us to study dynamical interactions across scales and quantify the trade-offs between such as localness or otherwise of energy transfer in the spectral space. We show that the total transfer rate is indeed local in wavenumber space, though the information about large scales is preserved by means of the low-frequency and low-amplitude fluctuations that traverse from low to high wavenumbers with increasing time lag.

4:53PM B28.00002 Emergence of turbulent behavior for passive scalars
DIEGO DONZIS, Texas A&M University, K.R. SREENIVASAN, NYU, V. YAKHOT, Boston University — We recently identified a transition to strong turbulence in isotropic turbulence with stochastic forcing. We found that the higher the order of the derivative moment the lower the transition Reynolds number. By a suitably rescaling, we could reduce the transition value to a universal Reynolds number of the order 10 for all orders. The moments of velocity gradients are Gaussian below this transition Reynolds number and exhibit anomalous scaling above, with the same scaling exponents as high-Reynolds-number turbulence. The matching of the two asymptotic states led to analytical expressions for scaling exponents in excellent agreement with available data. In this work we extend these concepts to passive scalar mixing. We observe a similar type of transition with the same Gaussian asymptotic behavior at low Reynolds numbers, but the transition Reynolds number depends on the Schmidt number (the ratio of viscosity to scalar diffusivity), and the matching of the two regimes yields scaling exponents that depend on the Schmidt number. The connection with the so-called ramp-cliff structures is also discussed. As for the velocity field, these results suggests that high-Reynolds-number behavior can be studied via well-resolved simulations around this low-Reynolds-number transition.

5:06PM B28.00003 Third-order structure functions for isotropic turbulence with bidirectional energy transfer
JIN-HAN XIE, College of Engineering, Peking University, OLIVER BUHLER, Courant Institute of Mathematical Sciences, New York University — We derive and test a new heuristic theory for third-order structure functions that resolve the forcing scale in the scenario of simultaneous spectral energy transfer to both small and large scales, which can occur naturally in rotating stratified turbulence or magnetohydrodynamical turbulence, for example. The theory has three parameters, namely the upscale/downscale energy transfer rates and the forcing scale, and it includes the classic inertial range theories as local limits. When applied to measured data, our global-in-scale theory can deduce the energy transfer rates using the full range of data, therefore it has broader applications compared with the local theories, especially in the situations where the data is imperfect. In addition, because of the resolution of forcing scales, the new theory can detect the scales of energy input, which was impossible before. We test our new theory with a two-dimensional simulation of MHD turbulence and use the theory to analysis geophysical fluid data.
5:19PM B28.00004 Inter-scale energy transfer by multiscale vorticity stretching and strain self-amplification in turbulence. PERRY JOHNSON, Stanford University — Three-dimensional turbulent flows are characterized by net transfer of energy from large to small scales. This inter-scale energy transfer is commonly described as a cascade driven by vorticity stretching, but in a phenomenological or imprecise way. Somewhat less commonly, the role of strain self-amplification is emphasized. This talk demonstrates an exact expression for inter-scale energy transfer in terms of multiscale vorticity stretching and strain self-amplification. This relationship elucidates the relative role of two of these two mechanisms in driving the cascade in the inertial range, while also accounting for the relative importance of scale-local and scale-nonlocal processes. Direct numerical simulations show that strain self-amplification contributes more to the energy cascade than vorticity stretching, but not overwhelmingly so. The leaky cascade view of inter-scale energy transfer is supported by the results. An additional mechanism of inter-scale energy transfer is revealed, with a possible connection to two-dimensional turbulence.

1This work was supported by the Advanced Simulation and Computing program of the US Department of Energy’s National Nuclear Security Administration via the PSAAP-II Center at Stanford, Grant No DE-NA0002373.

5:32PM B28.00005 Interscale Transport of the Reynolds Shear Stress in Wall-bounded Flows. RAMIS ORLÜ, Linné FLOW Centre, KTH Mechanics, GIOLE FERRANTE, ANDRES MORFIN, Università di Bologna, TAKUYA KAWATA, Tokyo University of Science, PHILLIP SCHLATTER, P. HENRIK ALFREDSSON, Linné FLOW Centre, KTH Mechanics — The interplay between the inner and outer layers in wall-bounded turbulent flows has become one of the focus areas with the advent of high-fidelity numerical and experimental data of sufficient scale separation, i.e. high enough Reynolds number. The general view is that the inner layer behaves autonomously in terms of its near-wall cycle and the outer layer exhibits independence of the details of the inner layer. At the same time, strict inner scaling for the near-wall region does not hold, due to the top-down influence from the large-scale structures further away from the wall. Recently, bottom-up influence (inverse energy transfer) has been observed, but primarily studied in terms of the turbulent kinetic energy. Here, using numerical simulation data from both a plane Couette flow and a turbulent boundary layer, the interscale transport of the Reynolds stresses is examined. Besides the classical interscale transport of turbulent kinetic energy towards smaller scales, also an inverse interscale transport of the Reynolds shear stress was observed. The spectral, scale-by-scale, analysis also indicates how interscale transport and turbulent diffusion explains the mismatch between the locations of the outer peaks in the Reynolds shear stress production and cospectrum.

5:45PM B28.00006 Impact of large-scale flow states on small-scale 3D turbulence. CRISTIAN C LALESCU, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization, Goettingen — Turbulent flows often feature an intricate interplay between anisotropic, even quasi-two-dimensional, large-scale features and three-dimensional small-scale fluctuations. To systematically study the relationship between large-scale flow structures and small-scale turbulence, we investigate a conceptually simple shear flow — a generalized turbulent Kolmogorov flow. The flow is forced with a single shear mode and is subject to large-scale friction, which effectively allows to control transitions between different large-scale states. We present a detailed investigation of the energetics of the system, and we find that the excitation of three-dimensional small-scale turbulence provides a direct kinetic energy cascade from large to small scales in the sense of classical eddy viscosity. We show that the energy transfer rate depends on the large-scale flow state, which establishes a direct coupling between the large scales and smaller-scale flow features such as small-scale intermittency.

1This project funded through the Swedish Research Council (VR)

5:58PM B28.00007 Energy flux vectors in two-dimensional anisotropic turbulence. MASANORI TAKAOKA, Department of Mechanical Engineering, Osaka University, TAKUYA KAWATA, Tokyo University of Science, PHILLIP SCHLATTER, P. HENRIK ALFREDSSON, Linné FLOW Centre, KTH Mechanics, GIOLE FERRANTE, ANDRES MORFIN, Università di Bologna, Bologna, TAKUYA KAWATA, Tokyo University of Science, PHILLIP SCHLATTER, P. HENRIK ALFREDSSON, Linné FLOW Centre, KTH Mechanics — The interplay between the inner and outer layers in wall-bounded turbulent flows has become one of the focus areas with the advent of high-fidelity numerical and experimental data of sufficient scale separation, i.e. high enough Reynolds number. The general view is that the inner layer behaves autonomously in terms of its near-wall cycle and the outer layer exhibits independence of the details of the inner layer. At the same time, strict inner scaling for the near-wall region does not hold, due to the top-down influence from the large-scale structures further away from the wall. Recently, bottom-up influence (inverse energy transfer) has been observed, but primarily studied in terms of the turbulent kinetic energy. Here, using numerical simulation data from both a plane Couette flow and a turbulent boundary layer, the interscale transport of the Reynolds stresses is examined. Besides the classical interscale transport of turbulent kinetic energy towards smaller scales, also an inverse interscale transport of the Reynolds shear stress was observed. The spectral, scale-by-scale, analysis also indicates how interscale transport and turbulent diffusion explains the mismatch between the locations of the outer peaks in the Reynolds shear stress production and cospectrum.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B29 Biological Fluid Dynamics : Respiratory Flows II

4:40PM B29.00001 A dynamical systems approach to particle transport in lung airways. ALI FARGHADAN, Northern Arizona University, FILIPPO COLETTI, University of Minnesota, AMIRHOSSEIN ARZANI, Northern Arizona University — Computer modeling of respiratory flows and particle transport are of both physiological and toxicological interests. In this talk, we present hidden dynamical systems features that control transport in human conducting airways. High-resolution computational fluid dynamics (CFD) simulation is carried out for an image-based tracheobronchial model under sinusoidal respiration flow. The destination map, which synchronizes the particle destination on the release point is found after performing Lagrangian particle tracking for microparticles. Finite-time Lyapunov exponent (FTLE) is calculated and inertial Lagrangian coherent structures (ILCS) are tracked during the breathing cycle. The results show that these dynamical systems features control the spatiotemporal evolution of the destination map at the trachea. Finally, slow manifolds are used as an efficient technique to identify the source of any arbitrary particle with backward integration of the Maxey-Riley equation. The novel dynamical systems techniques presented have important implications for drug delivery in respiratory disease.

JSPS KAKENHI Grant No. 18K03927, 19K03677
4:53PM B29.00002 Can we use CFD to improve targeted drug delivery in throat?1
SAIKAT BASU, South Dakota State University, RUPALI SHAH, ANDREW PAPPA, JIHONG WU, ALYSSA BURKE, WILLIAM BENNETT, WANDA BODNAR, JULIA KIMBELL, University of North Carolina at Chapel Hill — Numerical simulations of respiratory airflow and particle transport, along with synergistic physical experiments, can be used to identify the nebulized particle sizes that are most effective in enhancing targeted deposition at the laryngeal vocal fold granulomas in human throats. Narrow tracheal geometry results in high-speed inhaled airflow, leading to transitional and turbulent flow features. To account for short time-scale effects such as vortices, which can affect particle transport, our computational modeling scheme implements Large Eddy Simulations (LES) in three CT-derived anatomical mouth-nose-trachea reconstructions. To validate the numerical predictions, two distinct in vitro techniques, namely gamma scintigraphy and mass spectrophotometry, are used for measuring topical deposition in one CT-based solid model. Findings suggest a specific range \( \approx 8 – 10 \, \mu\text{m} \) of particle sizes with laryngeal granulomas and glottis as the specific deposition sites. The study considers three granuloma sizes (small, 3 mm; medium, 4.5 mm; large, 6 mm diameter) positioned at three distinct locations along the tissue lining of the vocal folds. The results have the potential to come up with novel personalized therapy protocol.

1Supported jointly by the National Center for Advancing Translational Sciences (NCATS), of the National Institutes of Health (NIH), through grant award UL1TR002489; and by the National Heart, Lung, and Blood Institute (NHLBI) of the NIH, under award R01HL122154

5:06PM B29.00003 Role of two-phase air-mucus interaction on aerosolized drug delivery in idealized lung models , RAHUL RAJENDRAN, ARINDAM BANERJEE, Lehigh University — Aerosolized drug delivery to the lung airways depends on the particle size, breathing pattern, and the airway geometry. The influence of the two-phase flow morphology and the local airflow structures developed in the mucus-lined airways of obstructive airway diseases, on the deposition of inhaled drugs is investigated. The Lagrangian particle tracking model is coupled with the multiphase Eulerian-Eulerian model to investigate the gas-liquid-solid flow in 3D idealized airway geometries. Mucus is modeled as a non-Newtonian fluid using the power-law equation, and flow turbulence is modeling using the low-Re \( k-\omega \) SST model, including the eddy interaction model (EIM) for the particles. The results from the study are validated with experimental results from the literature and comparisons with constricted dry-wall airways are presented. The role of mucus rheology, airflow rate, particle size, airway geometry and gravity are discussed. A quantitative assessment on the global and regional deposition fractions of the particles, sites of deposition, clearance rates, pressure drop across the airway, and airway resistance in mucus-lined airways will be presented.

5:19PM B29.00004 Effect of viscoelasticity and surfactant on an airway closure model1 , FRANCESCO ROMANO'. University of Michigan & ENSAM - ParisTech, HIDEKI FUJIOKA, Tulane University, METIN MURADOGLU, Koc University, JAMES B. GROTBERG, University of Michigan — A liquid made out of mucus and serous layers lines the human lung airways. Several different flow regimes are observed depending on the generation of the bifurcation and when small airways are considered (9th or 10th generation of the airway tree), surface tension effects dominate and they can induce a Plateau-Rayleigh instability. The airway is modeled as a rigid pipe coated with a single-layer fluid and the effects of surfactant and non-Newtonian properties of mucus are investigated using numerical simulations. The viscoelasticity of mucus is taken into account by means of the Oldroyd-B model, whereas surfactant is considered for a Newtonian lining fluid. The evolution equations of the interfacial and bulk surfactant concentrations are solved coupled with the incompressible Navier–Stokes equations. The viscoelastic behavior of mucus strongly increases post-coalescence wall shear stresses (about 50% in their peak value) when the Laplace number is high, whereas pre-coalescence stresses are almost insensitive to the Weissenberg number. Surfactant has a negligible effect on the closure-induced shear stress, except when the elasticity parameter or the surfactant concentration is high enough. Our parametric study covers both, healthy and pathological conditions.

1This work is supported by NIH grant HL136141 and by TUBITAK grant 115M688

5:32PM B29.00005 Effect of viscoelasticity and surfactant on the propagation and rupture of a liquid plug in an airway1 , METIN MURADOGLU, Koc University, FRANCESCO ROMANO'. University of Michigan & ENSAM - ParisTech, HIDEKI FUJIOKA, Tulane University, JAMES B. GROTBERG, University of Michigan — The propagation and rupture of a liquid plug in a distal airway are studied using numerical simulations. The simulations are carried out using a finite-difference/front-tracking method, previously validated for airway reopening with Newtonian fluids. The airway walls are considered rigid and the plug is driven by a pressure gradient enforced between the extrema of the pipe. The effect of interfacial and bulk surfactant is considered, together with the viscoelasticity of mucus, which is here taken into account using the FENE-P model. Our parametric study shows that the presence of surfactant can efficiently reduce the wall stresses along the airway wall, hence surfactant helps to reduce the damage on the epithelial cells distributed along the internal surface of an airway. Moreover, the effect of several viscoelastic parameters is considered, such as the Weissenberg number, the length of the polymer and the polymer-to-solvent viscosity ratio. Particular attention is paid to the distribution of extra stresses due to the non-Newtonian behavior of mucus since interesting elastic dynamics are triggered by the liquid plug rupture over the liquid-gas interface.

1This work is supported by NIH grant HL136141 and by TUBITAK grant 115M688

5:45PM B29.00006 Computational investigation of plug flow dynamics and splitting through 3D multi-branching bifurcating lung airway models , CORY HOI, University of Massachusetts, Dartmouth, ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — Liquid plug flow in capillary tubes has applications in medical procedures, including surfactant replacement therapy (SRT), which is used to treat respiratory distress syndrome in preterm infants by delivering surfactant plugs to their lungs. Current SRT procedures have a 35% non-response rate, which has been attributed to the complex fluid dynamics of liquid plug propagation through the lung airway network. Previous computational works performed 2D investigation of plug splitting in a single bifurcating airway geometry and mathematical models have been developed to calculate the plug split ratio at each independent airway bifurcation. In contrast, we present 3D CFD simulations of surfactant plug transport through multi-branching bifurcating lung airway models with three generations, in which upstream flows show a strong dependence on the downstream flow behavior of previously instilled plugs in subsequent airway generations, a phenomenon not captured in previous computational studies. Our simulations investigate the effects of plug instillation frequency, downstream plug blockages and plug rupture on plug split ratio and distribution, improving our understanding of SRT and helping to increase its effectiveness.
Accumulation of low-density lipoprotein (LDL) on the lumen surface of blood vessels is central to the initiation and progression of many cardiovascular diseases. It has been found that, in high Schmidt number flows, it is the near-wall flow dynamics that dictate the transport and accumulation of LDL. The mechanisms of fluid transport by cilia have been studied extensively at the level of the individual cilium and metachronal waves. However, how the topology of ciliary arrays is optimized to generate organ-scale directed flows is largely unexplored. Here, we image the mouse airway to map the geometry of its ciliary carpet, from the sub-cellular (10−6 m) to the organ scales (10−2 m), characterizing quantitatively its ciliary arrangement and the generated flows. Locally we measure heterogeneity in both cilia organization and flow structure, but across the trachea fluid transport is coherent. To examine this result, we develop a hydrodynamic model to explore systematically different tissue architectures. Surprisingly, we find that disorder enhances particle clearance, whether it originates from fluctuations, heterogeneity in multiciliated cell arrangement or ciliary misalignment. Together, our results shed light on how the microstructure of an active carpet determines its emergent dynamics and are applicable to understand airway pathologies.

1National Science Foundation Center for Cellular Construction (NSF grant DBI-1548297)

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B30 Biological Fluid Dynamics : Microfluidics 612 - C. Nadir Kaplan, Virginia Tech

4:40PM B30.00001 Thermal Effects on Fluid Mixing in the Eye1, JINGLIN HUANG, MORTEZA GHARIB, Caltech — Age-related macular degeneration (AMD) is the leading cause of central vision loss in the developed world. In the case of wet AMD, it can be managed through serial intravitreal injections of anti-vascular endothelial growth factor (anti-VEGF) agents. However, sometimes the treatment is ineffective and causes side effects. One possible cause of the ineffective treatment is the inefficient fluid mixing in the eye. Continued from my talk last year, we are now focusing on the understanding of thermal effects on fluid mixing in the vitreous chamber and various parameters that could affect it. The study outcomes will be useful for inspiring eye doctors to develop better strategies for improving treatment efficiency and optimizing patient experience.

1Acknowledgement: Sponsored by Chartrand Eye Research

4:53PM B30.00002 Computational analysis of interstitial fluid flow through the lacunar-canalicular system with morphological variations. SHAILESH KHADANGALE, SAMIRA HAJEBRAHIMI, MAUREEN LYNCH, DEBANJAN MUKHERJEE, University of Colorado Boulder — Osteocytes play a central role in maintenance of skeletal structure and associated mechanobiological processes. Interstitial fluid flow in the lacunar-canalicular system (LCS) is pivotal for osteocyte mechanotransduction but is challenging to model. For this we developed a CFD framework, based on Stokes-Brinkman model with variable permeability, for interstitial fluid flow in the LCS, and quantify shear at the osteocyte wall. We used our framework to quantify variations in shear due to changes in LCS morphology observed in metastatic bone cancer. A range of parametrically varied LCS morphologies were modeled based on lacuna dimensions and position data from micro-CT scan of healthy and cancerous mouse tibia. Shear on osteocytes was quantified for the various LCS morphologies considered to obtain bounds on osteocyte shear based on knowledge of lacuna shape and dimensions alone. Also, shear variations on osteocytes resulting from LCS morphology variations from neighboring sites were quantified. Our study revealed significant shear stress variations across all LCS morphological variations. Our over-arching theme is to advance this model into a computational toolkit to generate and test hypotheses on the role of shear mechano-transduction in metastatic bone cancer.

5:06PM B30.00003 Experimental modeling of fluid homeostasis in the mammalian hearing organ1, RUY IBANEZ, MOHAMMAD SHOKRIAN, JONG-HOON NAM, DOUGLAS H. KELLEY, University of Rochester — The mammalian hearing organ (cochlea) contains a long microfluidic channel (channel width ≈ 50 μm and aspect ratio ≈ 700). Ex vivo observations have shown that auditory stimulations induce deformations, in the form of a travelling wave, on the walls of the microfluidic channel and produce a flow. By determining the relevant physical parameters in the channel and applying scaling laws, we designed an apparatus that can replicate the physical conditions of the inner ear channel. We seek to characterize the induced flow using particle tracking velocimetry measurements, as well as characterizing the Lagrangian dynamics using a particle advection code. We validate the experimental measurements by comparing to previous analytic results. We study the effect of channels end boundary conditions (open or closed) and the shape of the wall deformation on the flow dynamics. We also find good agreement with finite-element simulations.

1NSF grant CMMI-1661413

5:19PM B30.00004 Prediction of Low-density Lipoprotein Concentration on the Luminal Surface of Pathological Blood Vessels Using Wall Shear Stress. SATYA JIT CHOUDHURY, Department of Applied Mechanics, Indian Institute of Technology Madras, KAMESWARARAO ANUPINDI, Department of Mechanical Engineering, Indian Institute of Technology Madras, B.S.V PATNAIK, Department of Applied Mechanics, Indian Institute of Technology Madras — Accumulation of low-density lipoprotein (LDL) on the lumen surface of blood vessels is central to the initiation and progression of many cardiovascular diseases. It has been found that, in high Schmidt number flows, it is the near-wall flow dynamics that dictate the transport and accumulation of the LDL on the lumen surface. Since wall shear stress (WSS) offers a reasonable approximation of near-wall flow dynamics, our study utilizes WSS to predict the variation of concentration of LDL on the lumen surface of symmetric 2D as well as asymmetric 3D pathological arteries. Blood is considered as Newtonian, incompressible and modeled using the Navier-Stokes equation whereas, the transport of LDL is governed by the passive scalar advection-diffusion equation. Even though the flow is pulsatile, it is found that time-averaged WSS gives a very good prediction of the variation of LDL concentration on the lumen surface. It is seen that low WSS need not necessarily lead to high LDL concentration. The influence of the stagnation points (region of zero WSS) on LDL concentration is also explored in the present work.
5:45PM B30.00006 Examining role of composition on the formation of extracellular polymeric substance (EPS) aggregates over a rising oil micro-droplet under shear. JIAN SHENG, MUN MUN NAHAR, ANDREW WHITE, MARYAM JALALI, Texas A&M. After Deepwater Horizon spill, it is believed that up to 15% of released oil settled to the sea floor as marine oil snow (MOS), which has been corroborated by field and laboratory observations. Several factors contributed to the production of MOS including particulate concentration and microbial mucous (e.g. EPS). EPS is a complex mixture of polysaccharides, proteins, nucleic acids and lipids, and their composition can vary significantly based on the microbial community and the environment. To examine quantitatively the role of EPS composition, specifically protein to hydrocarbon ratio (PHR), on aggregation, we use a microfluidics, “ecology-on-a-chip”, to simulate an rising oil drop through a suspension containing EPS and particulates. Time lapse microscopy lasting several days captures the growth and morphology of EPS aggregates. We demonstrate that EPS in the absence of particles is unable to form aggregates, while the addition of particles induces rapid aggregation. The higher PHR results in more stickiness of the EPS molecule and consequently leads to larger and more rapid formation of MOS. Ongoing work is considering the influence of flow shear and various EPS conformation on aggregate formation in flows. Funded by GoMRI & ARO.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B31 Biological Fluid Dynamics: Micro-swimmer Computational

4:40PM B31.00001 Instabilities and dynamics of phoretic suspensions1, TULLIO TRAVERSO2, SEBASTIEN MICHELIN, Ecole Polytechnique. Suspensions of Janus phoretic colloids are a canonical example of synthetic active fluids. Their potential applications range from technological to medical ones. Individual microscopic particles self-propel as a result of self-generated chemical gradients, and influence each other hydrodynamically and chemically. Such interactions lead to spontaneous nontrivial dynamics within phoretic suspensions, on length scales much larger than the swimmer size. We use a kinetic model to investigate the competition and interaction of self-propulsion with hydrodynamic and chemical couplings, whose characteristics are fundamentally determined by the shape and surface chemical properties of the particle, which are design parameters that can be optimized and controlled. Using a combination of linear stability analysis and nonlinear numerical simulations, we discuss the role of such design parameters in determining the onset of instabilities and subsequent nonlinear collective dynamics in dilute suspensions of chemically-active Janus swimmers.

1European Research Council
2Membership Pending

4:53PM B31.00002 The role of shape for a Brownian microswimmer interacting with walls. JEAN-LUC THIFFEAULT, HONGFEI CHEN, University Of Wisconsin - Madison. We consider a simple model of a two-dimensional microswimmer with fixed swimming speed. The direction of swimming changes according to a Brownian process, and the swimmer is interacting with boundaries. This is a standard model for a simple microswimmer, or a confined wormlike chain polymer. The shape of the swimmer determines the range of allowable values that its degrees of freedom can assume — its configuration space. Using natural assumptions about reflection of the swimmer at boundaries, we compute the swimmer’s invariant distribution across a channel consisting of two parallel walls, and the statistics of spreading in the longitudinal direction. This gives us the effective diffusion constant of the swimmer’s large scale motion. When the swimmer is longer than the channel width, it cannot reverse, and we then compute the mean drift velocity of the swimmer. This model offers insights into experiments of scattering of swimmers from boundaries, and serves as an exactly-solvable baseline when comparing to more complex models.

5:06PM B31.00003 Simulation and fabrication of neuromuscular biohybrid swimmers1, MATTIA GAZZOLA, ONUR AYDIN, XIAOTIAN ZHANG, TAHER SAIF, University of Illinois, Urbana-Champaign. Biohybrid machines have been developed using muscles to actuate soft robotic structures. The integration of neurons into the embodiment of such systems can transform them into intelligent machines able to adaptively respond to environmental cues. This relies on the ability of neural units to command muscle activity, making actuation through motor neurons the first milestone. Here, we achieve this milestone, and demonstrate neuromuscular actuation of a computationally designed biohybrid swimmer.

1NSF CAREER Grant No. CBET-1846752 (MG); NSF EFRI C3 SoRo Grant No. 1830881 (MG)
Reynolds number and elucidate the fundamental insights obtained. We expect our work to shed some light on the importance of stratification expenditure, hydrodynamic efficiency and the induced mixing by the swimmer. We explore this dependence in detail for both the regimes of field around the swimmer. This has a direct consequence on the motility characteristics of the swimmer such as swimming velocity, power or salinity. We divide our analysis in two regimes of low and high Reynolds numbers. We find that stratification significantly alters the flow study finds a direct application for swimmers in oceanic waters, where stratification occurs naturally either due to gradients in temperature or salinity. We divide our analysis in two regimes of fluid and elastic swimmer parameters when using noisy swimmer trajectory data at 30 Hz. This methodology can be used to develop artificial micro-swimmers and understand parameter ranges that allow for certain motility patterns.

According to the references, we can find insights into the locomotion of organisms living in such environments. For instance, Jeffery (1922) and Gustavsson and Mehlig (2016) provide foundational work in this area. These studies have implications for both plankton ecology and medical applications, where precise control over swimming behavior is crucial. Direct numerical simulations (DNS) can be used to integrate the Lagrangian trajectories of particles swimming in turbulence with fixed speed and orientation governed by fluid gradients. Swimmers elongated along their swimming axis, align with the local flow velocity, with preferential downstream swimming. By the perturbative solution of a statistical model, we can show how the alignment is due to the peculiar correlation of fluid velocity and its gradients along particle paths caused by swimming. Numerical computation of the relevant correlations in DNS results shows that the theoretical prediction applies with remarkable precision to turbulent flows.

**References**

1. M Borgnino, et al. (Submitted to Phys Rev Lett, 2019)
In silico modeling of formation and growth of thrombus under blood flow.

4:53PM B32.00002
NIKHIL JANARDAN YEWALE, SATYAJIT CHOUDHURY, B.S.V PATNAIK, Department of Applied Mechanics, Indian Institute of Technology Madras. — Understanding the spatio-temporal evolution of thrombus — biochemical species is critical to simulate the initiation and propagation of pathways that lead to thrombus formation in blood vessels. A large number of species and their reactions are challenging to synthesize through the experimental means. Thus, in silico modeling of the blood clot formation serves as a useful tool. To this end, blood is assumed as a multi-constituent mixture comprising of fluid and thrombus phase, which transports various biochemical agonists and inhibitors contributing to the coagulation cascades. In this study, blood is modeled using source term to account for the resistance on the blood flow from the thrombus formed. The transport of biochemical species involved is represented with convection-diffusion-reaction equations. The injury site is subjected to a constant influx of chemical agonist. The model used in this study accounts for the embolization of clot due to shear stress. We present the CFD simulation of the growth of thrombus at the site of endothelial injury due to chemical and shear-induced activation of platelets in a straight and stenosed blood vessel. The study also highlights the influence of flow pulsatility on the growth of thrombus.

5:06PM B32.00003
On cell proliferation in a tissue engineering scaffold pore, effects of nutrient concentration and scaffold internal geometry.

ZESHUN ZONG, Courant Institute, New York University, XINYU LI, New York University, Shanghai, PEJIAN SANAEI, New York Institute of Technology. — Cell proliferation within a porous tissue engineering scaffold perfused with nutrient solution depends sensitively on the choice of pore geometry, flow rates, and nutrient concentration. Regions of high pore curvature encourage cell proliferation, while a critical flow rate is required to promote growth. Moreover, the dynamics of the nutrient culture medium consumption influence the cell growth. In experiments, such factors should be chosen meticulously to match the characteristics of the underlying cells and the particular goal of incubation. However, determining these factors poses a significant challenge that cannot be addressed by experimentation alone. In this talk, we present the first-principle mathematical theory for the nutrient concentration coupled to the growth of cells seeded on the pore walls, which is driven by the fluid flow within a tissue engineering scaffold pore. In addition, using asymptotic analysis based on the pore small aspect ratio, we derive a reduced model that enables a comprehensive analysis of the system to be performed. This approach reduces the number of parameters, captures the experimental observations and suggests improvements to the design of a tissue engineering scaffold and the appropriate operating regime.

The authors gratefully acknowledge support from the following sources: P.S. from the NSF, RTG/DMS-1646339. X.L. from the SURE offered by Courant Institute of Mathematical Sciences, New York University.

5:19PM B32.00004
An Assessment of Thrust, Drag, and Momentum Exchange of Undulation-Based Propulsion.

GEORGE LOUBIMOV, MICHAEL KINZEL, University of Central Florida. — Studies have shown increases in efficiency for undulating propulsion through interactions with unsteady wakes. Specifically, performance gains are related to favorable interactions between an undulation-like swimmer and an oncoming, unsteady wake such as that of a schooling fish. In this study, Computational Fluid Dynamics (CFD) is used to evaluate the unsteady fluid interactions associated with undulation-based propulsion. The numerical accuracy of the CFD model is established and also shown to correlate well with benchmark experiments. While a number of methods have been used to design an undulation-based swimmer, the above approach is based on the use of unsteady wakes, the fundamental physics of the fluid mechanics responsible for propulsion is not fully understood. The aim of the effort is to refine the understanding of the forces associated with the unsteady wakes on an undulating foil. It is proposed that through evaluating the total pressure changes, shear force, and volume momentum changes during these interactions, additional insight can be developed. Using this approach, we believe we can identify the key fluid criteria responsible for increasing the propulsive efficiency of undulating swimmers.

5:32PM B32.00005
Pumping at low Reynolds numbers - the leucon sponge pump.

JENS HONORE WALTHER, SAYED SAED ASADZADEH, POUL SCHEEL LARSEN, Technical University of Denmark, HANS ULRIK RISGRD, University of Southern Denmark. — Leucon sponges are filter-feeders with a complex system of branching inhalant and exhalant canals leading to and from the close-packed choanocyte chambers. Each of these choanocyte chambers holds many choanocytes that act as pumping units delivering the relatively high pressure rise needed to overcome the system pressure losses in canals and constrictions. We study these pumping units by solving the Navier-Stokes equations using computational fluid dynamics simulations. We find that each choanocyte acts as a leaky, positive displacement pump working at the interaction between its beating flagellum and the collar, open at the base for inflow but sealed above. The leaking backflow is caused by small gaps between the vaned flagellum and the collar. The choanocyte pumps act in parallel, each delivering the same high pressure, because low-pressure and high-pressure zones in the choanocyte chamber are separated by a seal – secondary reticulum. The mechanical pump power expended by the beating flagellum is compared with the useful (reversible) pumping power received by the water flow to arrive at a typical mechanical pump efficiency of about 70%.

This work was supported by a research grant no. (9278) from VILLUM FONDEN.

5:45PM B32.00006
Fluid Dynamics of Ballistic Strategies in Nematocyst Firing.

CHRISTINA HAMLET, Bucknell University, WANDA STRYCHALSKI, Case Western Reserve University, LAURA MILLER, University of North Carolina at Chapel Hill. — Nematocysts are stinging organs used by members of the phylum Cnidaria (jellyfish, anemones, hydra) for capturing prey and other important functions. Nematocysts are some of the fastest-known accelerating structures in the animal world. As such their rapid accelerations and small scales complicate resolving some aspects of their firing mechanism. We present results from mathematical models implemented in an immersed boundary framework that capture some of the dynamics of a a Barb-like structure accelerating a short distance across a range of Reynolds numbers toward a passive target. Results indicate that acceleration and then coasting is not sufficient for a nematocyst to reach its target. We discuss the implications of these results for mechanisms required for small-scale ballistics.

5:58PM B32.00007
Growth and adaptation in a fungal hydraulic network.

BOHYUN KIM, UCLA Math, JOSEPH BECKMANN COLLABORATION, MATTHEW CHEUNG COLLABORATION, JOHN LENTFER COLLABORATION, RAMONA SASSE COLLABORATION. — Biological networks such as fungal hydraulic networks have evolved to solve many of the same problems as human-built transportation networks. However, they must also briskly adapt to changing environments and modify their architecture without centralized control. Working with the model filamentous fungus, Neurospora crassa, we characterize the growth of the network and propose a mathematical model for the collaborative behavior of its cells at various scales. We compare our model with measurements of the complex distribution of flows and resources across the cellular network.

Saturday, November 23, 2019 4:40PM - 5:45PM
Session B33 Flow Instability: Viscoelastics
615 - Marie-Charlotte Renoult, INSA de Ronen
4:40PM B33.00001 Experimental investigation Faraday wave onset in viscoelastic materials\textsuperscript{1}. XINGCHEN SHAO, J. R. SAYLOR, JOSHUA BOSTWICK, Clemson University, PASQUALE CIARLETTA, Politecnico di Milano — Herein we explored the onset of parametrically excited surface waves (Faraday waves) on glycerin-water mixtures and agarose gels, ascertaining the effect of viscosity and elasticity on the threshold amplitude and mode selection. Faraday waves were mechanically generated in a circular tank mounted on an electromechanical shaker over a range of driving frequencies. We obtained multiple Faraday instability modes, each of which is characterized by a radial and azimuthal wave number and by a unique stability tongue in the amplitude-frequency space. Glycerin/water mixtures were used to explore a range of viscosities showing that for a given mode the frequency at onset decreases with viscosity, while the amplitude increases. In contrast, for gels, the onset frequency and amplitude both increase with elasticity. Experiments are then compared against theory. This work could potentially be used to develop a diagnostic method for measuring the complex modulus of viscoelastic materials.

\textsuperscript{1}Funding support from NSF Grant CBET-1750208 is acknowledged

4:53PM B33.00002 Viscoelastic flow instabilities in static mixers: onset and effects on the mixing efficiency. SIMONA MIGLIOZZI, GIOVANNI MERIDIANO, LUCA MAZZEI, PANAGIOTA ANGELI, Dept. of Chemical Engineering, University College London, CORAL COLLABORATION — Purely elastic instabilities occur in the absence of inertial effects, induced by the combination of strong elastic forces with nonlinearities of the flow. In a laminar mixing process, the onset of these instabilities is likely to occur in the window of applied flow rates, therefore it is of paramount importance to understand the effects of their onset on the mixing efficiency. In this work, we experimentally investigate the onset of elastic instabilities in two in-line mixers with different geometric features, i.e. a Kenics helical mixer and a GFX mixer, characterised by a double X-shaped bars geometry. Concentrations maps were obtained at different mixer lengths by means of Planar Laser Induced Fluorescence. We mapped the onset of the instabilities with Reynolds and Deborah numbers. Three fluids with different rheological behaviour – i.e. a Boger fluid and two shear-thinning fluids – were tested to deduced a generalised effect of the fluid elasticity on the mixing patterns. The effect of the instabilities depended on the different kinematics induced by the two distinct geometries: for the helical mixer the typical lamellar structure is not recovered and the two liquid streams remain unmixed, while for the GFX mixer the concentration maps oscillate in time in a quasi-periodic fashion. In both cases, the onset of instabilities worsens the mixing efficiency with respect to the Newtonian case.

5:06PM B33.00003 Three-dimensional viscoelastic flow instabilities at extensional stagnation points. BECCA THOMASES, PALOMA GUTIERREZ, ADAM KAGEL, University of California, Davis — Simulations of viscoelastic fluid models in the Stokes limit exhibit instabilities at extensional stagnation points. We present simulations of both the Oldroyd-B and FENE-P models in a 3D periodic domain with cylindrical 4-roll mill background forcing. Beyond a critical Weissenberg number (non-dimensional relaxation time) we find an instability in the z-direction (note that the background force is constant in z). We present both simulations and linear stability analysis and identify criteria for the occurrence of this instability.

5:19PM B33.00004 Proper orthogonal decomposition of viscoelastic liquid jets. LOUISE COTTIER, MARIE-CHARLOTTE RENOUlt, CORIA-UMR 6614 INSA Rouen Normandy, CHRISTOPHE DUMOUCHEL, CORIA-UMR 6614 CNRS — Proper Orthogonal Decomposition (POD) is a linear procedure that allows to identify characteristics of a data set by determining an optimized set of function basis. The optimization of that new basis is achieved by maximizing the projection of the data set on it. POD therefore depends on the choice of the scalar product. In the field of fluid dynamics, POD has been mostly applied to single phase turbulent flows, using the classical Euclidean scalar product. Here, we apply POD to a two-phase laminar flow: a low-velocity viscoelastic liquid jet evolving in an inert gas. More precisely, the procedure is applied to the interface between the two phases, and the scalar product is sought in the aim of tracking the jet surface evolution. After a brief presentation of the POD concept, we will describe the operating of our POD procedure on viscoelastic liquid jet images obtained from our experiments. Then, the results regarding pattern identification will be exposed. Finally, we will state what kind of physical information POD could bring about an atomization process involving viscoelasticity.

5:32PM B33.00005 Analysis for inertial and elastic instabilities in extensional flow and comparisons with cross-slot flow. HOWARD HU, RANJIANGSHANG RAN, PAULO ARRATIA, University of Pennsylvania, H. HU, R. RAN AND P. ARRATIA TEAM — We theoretically investigate the instabilities of a steady planar extensional flow of viscoelastic fluids with the flow vorticity equation. The results of this linear stability analysis indicate two distinct instabilities depending on the values of Reynolds number (Re) and Weissenberg number (Wi). One instability is an inertia-dominated one occurring at a critical Re, in which the vorticity component $\omega_z$ becomes unstable, suggesting an emerging axial vortex in the extensional direction $x$. The other instability is an elasticity-dominated one at high Wi, where the vorticity component $\omega_z$ in the direction normal to elongational plane becomes unstable, indicating a symmetry breaking on the elongational $xy$-plane. The predicted critical $Re$ and $Wi$ numbers of these two instabilities by the linear stability theory are critically compared with experimental and numerical results in the cross-slot channel flows.

\textbf{Saturday, November 23, 2019 4:40PM - 6:11PM – Session B34 Flow Instability: Boundary Layers Transition 616 - Tim Colonius, Caltech}

4:40PM B34.00001 Volume-Area and Area-Perimeter Dimensions of Turbulent-Spots Interfaces in Transitional Boundary-Layer Flow\textsuperscript{1}. ZHAO WU, TAMER ZAKI, CHARLES MEN-EVEAU, Johns Hopkins University — The nature of turbulent spots in transitional boundary layers, and whether their internal structure shares characteristics of equilibrium turbulence, remain open questions of considerable interest. Here we study scaling properties of the interface separating the spots from the outside flow. For high-Reynolds number turbulence, the outer interfaces are known to display fractal scaling with a fractal dimension near $D=2+1/3$, where the 1/3 can be related to the Kolmogorov scaling of velocity fluctuations (e.g. de Silva et al. PRL 2013). We measure the volume-area fractal scaling of the naturally triggered turbulent spots. The data are from the DNS of a transitional boundary layer available at the JHTDB (http://turbulence.pha.jhu.edu). The spot boundaries (interfaces) are determined without arbitrary threshold selection using an unsupervised machine learning method, namely the self-organizing map (Zhao et al. PRF 2019). Results from the volume-area fractal dimension confirm $D=7/3$, i.e. trends consistent with fully developed turbulence. Applying an alternative area-perimeter analysis on planar cuts at various heights shows D decreasing then increasing. It is argued that these trends could be associated to changes in the thickness of the interface at different heights from the wall.

\textsuperscript{1}National Science Foundation (grant \# OCE-1633124) and Office of Naval Research (grant No. N00014-17-1-2937).
4:53PM B34.00002 On the evolution of the velocity gradient in a minimal simulation unit of transitional boundary layers\textsuperscript{1} , AHMED ELNAHHAS, PERRY JOHNSON, ADRIAN LOZANO-DURAN, PARVIZ MOIN, Center for Turbulence Research, Stanford University | The transition of a boundary layer from laminar to turbulent state is associated with a rapid increase in friction and heat transfer coefficients and is accompanied by the rapid growth of velocity gradients throughout the boundary layer. This can be appreciated by visualizing isosurfaces of Q-criterion or other vortex identifiers and is particularly evident in late-stage transition when growing structures abruptly break down into more chaotic flow, generating turbulent spots. We consider the evolution of several velocity gradient invariants integrated in cross-stream planes using direct numerical simulations of canonical transition scenarios with minimal spanwise extent. The spanwise domain is restricted to fit only one wavelength of the oblique wave, leading to a single \( \Lambda \)-vortex being present at any streamwise location. The minimal unit simulation displays the main features of transition in larger domains, such as a realistic skin friction coefficient profile. The budget of the squared Frobenius norm of the velocity gradient tensor shows that the pressure term is orders of magnitude smaller compared to the source and sink terms. Furthermore, the profile of the Q-criterion squared exhibits a distinct plateau after the initial emergence of the hairpin vortex, and appears to be a good indicator of the onset of transition.

\textsuperscript{1}Supported by NASA and DOE

5:06PM B34.00003 Development for a Theoretical Model of Crossflow-induced Boundary-layer Transition\textsuperscript{1} , MAKOTO HIROTA, Institute of Fluid Science, Tohoku University, YUKI IDE, Japan Aerospace Exploration Agency, TAKAHISA HAYASHIDA, Graduate School of Information Sciences, Tohoku University, YUII HATTORI, Institute of Fluid Science, Tohoku University | On widely-used swept wings of aircrafts, laminar-turbulent transition of three-dimensional boundary layer mainly occurs through the process in which (i) a crossflow instability (referred to as the primary) first grows spatially and generates a vortex street and, then, (ii) the vortex street further induces low-speed streaks in the mainstream distribution that becomes unstable to a high-frequency secondary instability. Direct numerical simulation (DNS) can reproduce this process accurately, but the transition location sensitively depends on how the two kind of disturbance sources are fed to the primary and secondary instabilities, respectively. It is moreover difficult to understand the dependency on various flow parameters. In this study, we develop a theoretical model to estimate the growth rates of both the primary and secondary instabilities according to linear stability analyses. By noting a scale similarity to the Kelvin-Helmholtz instability, the inflection point and the shear profile of the flow enable us to estimate the growth rates. Our method is expected to be not only efficient to predict the transition location, but also useful for finding less unstable flow profiles that control devices should attain.

\textsuperscript{1}This work is based on results obtained from a project commissioned by the New Energy and Industrial Technology Development Organization (NEDO).

5:19PM B34.00004 Emergence of streaks and turbulent spots in an unsteady boundary layer beneath a solitary wave , ASIM ONDER, PHILIP LI-FAN LIU, National University of Singapore | Bypass route to transition is studied in a bottom boundary layer developing under solitary wave. First, the conditions for streak growth and breakdown are analyzed using a linear input-output framework and secondary stability analysis. Vortical perturbations whose intensity is about 1% of the maximum free-stream velocity are found to be sufficient to induce unstable streaks in moderate to high Reynolds numbers. In the second step, a natural bypass transition scenario is realized using direct numerical simulations, where a weak turbulent current is introduced to initiate the transition. The breakdown of streaks to turbulent spots is shown. Depending on their nucleation phase, the turbulent spots can grow to occupy the whole domain leading to a premature transition bypassing the emergence of two-dimensional modal instabilities.

5:32PM B34.00005 Linear and nonlinear dynamics of second-mode instability in hypersonic boundary layers (HBL)\textsuperscript{1} , UNNIKRISHNAN SASIDHARAN NAIR, DATTA GAITONDE, The Ohio State University | Hypersonic transition is often dominated by the second-mode instability. We perform a direct numerical simulation (DNS) informed by linear stability theory, to understand the eventual three-dimensional breakdown of this instability. Linear amplification and nonlinear saturation of this two-dimensional wave eventually culminate in the appearance of oblique waves, which break down the HBL into a turbulent state. A modal analysis of the transitional region identifies lambda-shaped vortices belonging to both fundamental and subharmonic categories. While the former appears relatively downstream, the subharmonic waves are observed immediately following nonlinear saturation. This nonlinear stage of transition is further analyzed through a novel unsteady flow perturbation (UFP) technique. UFP essentially tracks the linear evolution of perturbations on a nonlinearly saturated background flow, thus approximating a Floquet analysis for general configurations. UFP is shown to identify the most receptive superharmonic/subharmonic components in the periodically distorted flow. In addition to providing insights into the dynamics, relative to DNS, it provides an accurate low cost approximation of the breakdown spectrum in the early transitional stages.

\textsuperscript{1}Office of Naval Research (Grants: N0001-13-1-0534, N00014-17-1-2584)

5:45PM B34.00006 Nonlinear input-output analysis of laminar-turbulent transition for wall-bounded flows\textsuperscript{1} , GEORGIOS RIGAS, Imperial College London, DENIS SIPP, ONERA, TIM COLONIUS, Caltech | In a linear input-output analysis framework, the most amplified instabilities are typically described by considering singular vectors of the resolvent operator of the linearized Navier-Stokes equations. In this study, we extend the methodology to take into account nonlinear triadic interactions by considering a finite number of harmonics in the frequency domain using the Harmonic Balance Method. Optimal nonlinear forcing mechanisms that lead to transition and maximize the skin-friction coefficient are identified using direct-adjoint looping. We demonstrate the framework on a zero-pressure flat-plate boundary layer considering three-dimensional perturbations triggered by a few optimal forcing modes of finite amplitude. Depending on the frequency, spanwise wavenumber, amplitude and symmetries of the perturbation, we recover all the transition stages associated with K-type and H-type transition mechanisms, oblique waves, streaks, and their breakdown. The proposed frequency-domain framework identifies the worst-case frequency disturbances forwall-bounded laminar-turbulent transition.

\textsuperscript{1}G.R. and T.C. acknowledge support from The Boeing Company (CT-BA-GTA-1)
5:58PM B34.00007 The Role of Fluctuating Dissipative Fluxes in the Receptivity of High-Speed Chemically Reacting Boundary Layers in Binary Mixtures to Kinetic Fluctuations

Asymptotic approximations are derived for the hydrodynamic force on a rigid, axisymmetric particle executing longitudinal or transverse oscillation in unsteady Stokes flow. The slender particle has an aspect ratio $a/L << 1$, where $L$ is the half-length of the particle, and $a$ is its characteristic cross-sectional width. The frequency of oscillation is parameterized by the complex quantity $\lambda^2 = -i\omega^2\nu/\nu$, where $\nu$ is kinematic viscosity, $\omega$ is particle oscillation frequency, and $i = \sqrt{-1}$. Asymptotic approximations for the force are obtained in three frequency regimes: (i) low frequency, $\lambda << 1$; (ii) moderate frequency, $\lambda \sim O(1)$; and (iii) high frequency, $\lambda >> 1$. Physical interpretations of the force in each regime are made and compared between the longitudinal and transverse oscillation cases. Our asymptotic predictions are in good agreement with the numerically computed frequency-dependent force on a prolate spheroid ($\epsilon = 0.1$) for longitudinal and transverse oscillations by Lawrence and Weinbaum (J. Fluid Mech., vol. 189, 1988) and Pozrikidis (Phys. Fluids, vol. 1, 1989), respectively.

Saturday, November 23, 2019 4:40PM - 6:11PM –
Session B35 Microscale Flows: Oscillations 617 - Henry Chu, Carnegie Mellon

4:40PM B35.00001 The force on a slender particle under oscillatory translational motion in unsteady Stokes flow

Asymptotic approximations are derived for the hydrodynamic force on a rigid, axisymmetric particle executing longitudinal or transverse oscillation in unsteady Stokes flow. The slender particle has an aspect ratio $a/L << 1$, where $L$ is the half-length of the particle, and $a$ is its characteristic cross-sectional width. The frequency of oscillation is parameterized by the complex quantity $\lambda^2 = -i\omega^2\nu/\nu$, where $\nu$ is kinematic viscosity, $\omega$ is particle oscillation frequency, and $i = \sqrt{-1}$. Asymptotic approximations for the force are obtained in three frequency regimes: (i) low frequency, $\lambda << 1$; (ii) moderate frequency, $\lambda \sim O(1)$; and (iii) high frequency, $\lambda >> 1$. Physical interpretations of the force in each regime are made and compared between the longitudinal and transverse oscillation cases. Our asymptotic predictions are in good agreement with the numerically computed frequency-dependent force on a prolate spheroid ($\epsilon = 0.1$) for longitudinal and transverse oscillations by Lawrence and Weinbaum (J. Fluid Mech., vol. 189, 1988) and Pozrikidis (Phys. Fluids, vol. 1, 1989), respectively.

1. J. K. K. acknowledges financial support from the Air Products Graduate Fellowship and the Sharbaugh Presidential Fellowship

4:53PM B35.00002 Rectified inertial forces in oscillatory flows: theory, simulation, experiment

Theory, simulation, and experiment show that inertial forces in oscillatory flows can rectify fluid motion, leading to effective flow rectification even in fluids where viscosity dominates. This phenomenon is important in a wide range of applications, from microfluidics to astrophysics. Theoretical predictions are confirmed through numerical simulations and experiments, highlighting the potential of oscillatory flows for flow control and propulsion in microscale devices.

5:06PM B35.00003 A simple and effect method for generating sub-kilohertz oscillatory flows in microchannels

By exploiting the physics of oscillatory flow, a simple and effective method for generating sub-kilohertz oscillatory flows in microchannels is developed. This method is applicable to a wide range of microfluidic devices and can be used for applications such as mixing, trapping, and propulsion in microscale systems.

5:19PM B35.00004 New insights into the migration patterns of a single capsule flowing in a pulsating microchannel

Using advanced imaging techniques, new insights into the migration patterns of a single capsule flowing in a pulsating microchannel are revealed. The results challenge traditional assumptions about capsule behavior and provide a deeper understanding of the complex dynamics involved in such systems.
Continuum breakdown of the acoustic field generated by a pulsating micro-sphere\(^1\), YARON BEN-AMI, AVSHALOM MANELA, Technion - Israel Institute of Technology — The flow field of a pulsating sphere is a canonical problem in acoustics. We consider the counterpart problem at small length scales of the sphere, where its radius is comparable with the gas mean free path. Such small scales are often encountered in the process of time-resolved spectroscopy of micro-particles. At these conditions, the gas rarefaction effects become important and the continuum description breaks down. The acoustic field is analyzed in the entire range of sphere length-scales. Numerical calculations are carried out via the direct simulation Monte Carlo method; analytical predictions are obtained in the free-molecular and near-continuum regimes. In the latter, the regularized thirteen moments model is applied, to capture the system response at states where the Navier-Stokes-Fourier equations break down. The results quantitate the dampening effect of gas rarefaction on the acoustic field. At near-continuum conditions, the acoustic field is composed of exponentially decaying ‘compression’, ‘thermal’ and ‘Knudsen-layer’ modes, reflecting thermoviscous and higher-order rarefaction effects. Stronger attenuation is obtained in the free-molecular regime, where boundary sphericity results in a geometric reduction of the molecular layer affected by the source.

\(^1\)This research was supported by the Israel Science Foundation (grant no. 1084/16) and the Aeronautical Engineering Research Fund. Y.B. acknowledges the support by the Adams Fellowship Program of the Israel Academy of Sciences and Humanities.

Application of Nonlinear Electrokinetic Transport using Network Heterogeneity of Porous Media\(^1\), HYEKYUNG LEE, Seoul National University, ALI MANI, Stanford University, SUNG JAE KIM, Seoul National University — Nanoscale electrokinetic transport through perm-selective membranes has been actively researched using a micro/nanofluidic platform recently. The ion depletion layer formed near the membrane under dc bias can be compartmentalized using heterogeneous microchannels and a numerical/experimental study using the simplified porous media demonstrated an internal recirculation flow due to different hydraulic resistance in network of channels. Furthermore, overlimiting current significantly increased due to this flow so that the channel with the higher resistance played as the main current path. In this presentation, we applied this mechanism to practical engineering problems. First, we investigated electrokinetics by laying a permselective membrane on only the main current path. By comparing the devices either with uniformly patterned membrane vs non-uniformly patterned membrane (i.e. membrane only at the current paths), a similar perm-selective ion transportation was measured, leading to the same mass transfer but the half of membrane material cost. Second, it is expected that network effect could localize the membrane fouling because the mass transfer is dominant through the main current path.

\(^1\)This work is supported by Basic Research Laboratory Project (NRF-2018R1A4A1022513) by the Ministry of Science and ICT, Korea, BK21 plus program in SNU.

Capacitive Charging of Multi-ion Electrolytes confined in parallel electrodes\(^1\), YUN SUNG PARK, IN SEOK KANG, Department of Chemical Engineering, POSTECH — Charging and discharging of ionic devices have been widely studied theoretically and experimentally. Especially, numerous kinds of electrolytes and their combinations have been utilized in order to enhance capacitances or frequency dependencies of the devices. However, the theoretical studies of effects of multiple ions on the properties of the devices are insufficient. In this work, we study the effect of multiple ions quantitatively using continuum approach. Electrolytes are confined in one-dimensional parallel electrodes, which is simplest system for many ionic devices. The multiple ions with different sizes and valences are chosen and their effects are compared along with the sparse and dense bulk concentrations. We focus on both the equilibrium distributions and the dynamic behaviors of ions, which are related to capacitances and frequency dependencies, respectively. The quantitative analyses are done numerically by dividing charge densities and fluxes of individual ions. Lastly, simple electroosmotic flows are calculated from the obtained charge densities.

\(^1\)This research was supported by the National Research Foundation of Korea (NRF) Grant funded by the Korea Government (MSIP: Ministry of Science, ICT & Future Planning) (No.2017R1D1A1B05035211).

Negative Pressure of Ionic Liquids Inside a Nanoslit: Molecular Dynamics Study\(^1\), YU DONG YANG, JUNG MIN OH, IN SEOK KANG, POSTECH — Predicting the force the acting on the charged surface is important to prevent deformation or swelling of pores. When the electrostatic interaction between charged pore and its counter-ions is predominant, the internal electrolyte can pull the wall. This means the total pressure becomes negative. Since it becomes complicated to predict with considering the large size and complex shape of ionic liquids, molecular dynamics simulations are conducted to analyze the pressure acting on the nanoslit wall. The pressure change with respect to the slit width under 1 nm scale was analyzed using coarse-grained model, and it is determined by the competition between the contact and electrostatic component of the total pressure. When the slit width is below a double of each ion size, the negative pressure scale is about \(10^3\) atm and it is 20 to 40 times the bulk osmotic pressure of each ionic liquids. The ion shape or size is related to the magnitude of contact component, but it hardly affects the negative pressure when the electrostatic component is dominant due to the ion re-arrangement in a slit.

\(^1\)This work is supported by the National Research Foundation of Korea (No. 2017R1D1A1B05035211) and BK 21 Plus program of Korea.
5:19PM B36.00004 Understanding the electrokinetic interaction between dynamically tunable nanofluidic channels connected in series. BARAK SABBAGH, Technion - Israel Institute of Technology, ELAD STOLOVIC, Harvard University, GILAD YOSSIFON, Technion - Israel Institute of Technology. The passage of an electric current through a permselective medium results in a phenomenon known as concentration polarization (CP). This phenomenon has been the focus of intense research, particularly regarding its relationship with microfluidic applications, e.g., on-chip desalination and enhanced biosensing sensitivity. These effects have been observed in a fixed geometry and properties of the permselective medium (i.e., nanochannel) which defines the location, intensity, and length of the CP layer. The ability to dynamically tune the geometry of the permselective medium allows us to optimize existing processes and open up new application possibilities. This is realized using pneumatically controlled microvalves where the formed gap between the elastically deformed membrane, which is controlled by external pressure chamber, and the nanochannel, forms the ion permselective nanochannel. In contrast to previous studies of such a system, here we extend this approach to several individually addressable microchannels that are connected in series within the same microchannel. Such interaction between two or more dynamically formed nanochannels yields interesting results of both the transient and steady-state behavior of the CP layers and related electrical response.

5:32PM B36.00005 Revisiting, resolving and unifying the nanochannel-microchannel electrical resistance paradigm. RAMADAN ABU-RJAL, RAN ESHEL, Technion-Israel Institute of Technology, YOAV GREEN, Ben-Gurion University of the Negev. Until recently, the accepted paradigm was that of Ohmic electrical response of nanochannel-microchannel systems determined solely by the nanochannel while the effects of the adjacent microchannels are negligible. Two, almost identical, models were suggested to rationalize experimental observations that appeared to confirm the paradigm. However, recent works have challenged this paradigm and showed it to be incorrect, namely, the microchannels contribute in a non-negligible manner. Two newer nanochannel-microchannel models were suggested to replace the nanochannel-only models. These models were asymptotic solutions limited to either very low or very high concentrations. Here, we review these four leading models. The most popular is shown to be incorrect, while the remaining models are unified under a newly derived solution which shows remarkable correspondence to simulations and experiments. The unifying model can be used to improve the design of any nanofluidic based systems as the physics are more transparent, and the need for complicated time-consuming preliminary simulations and experiments has been eliminated.

5:45PM B36.00006 Enhanced Electrokinetic Energy Conversion & Ion-Selective Transport in Macroscopic Vertically Aligned BNNT Membranes. SEMIH CETINDAG, Rutgers University, AADITYA PENDSE, University of Illinois, Chicago, ROBERT F. PRAINO, Chasm Advanced Materials, SANGIL KIM, University of Illinois, Chicago, JERRY W. SHAN, Rutgers University. Recent nanofluidic experiments with single or few nanopenes in graphene, MoS2, and h-BN have shown unique fluidic transport properties and the potential for electrokinetic energy conversion with unprecedented power densities. In such nanopenes, the high-surface charge makes possible a diffusio-osmotic mechanism for ion-selective transport, distinct from the Donnan exclusion in the conventional membranes. Here, we describe the fabrication of the first-ever macroscopic vertically aligned boron-nitride nanotube (VA-BNNT) membranes, and our study of their ion-selectivity mechanisms and osmotic-power-generation performance. We show that the VA-BNNT membranes are highly cation-selective even when the Debye length is smaller than the inner-pore radius of the nanotubes. Moreover, the membranes exhibit osmotic-energy-conversion efficiencies of 30%, and have osmotic power densities (based on open pore area) comparable to and even exceeding that of single BNNTs, up to 7,500 W/m² at pH 11 for a 1M:1mM KCl molarity difference. This osmotic power density increase with increasing surface-charge density at higher pH, but remain substantial even at pH 7. To further elucidate the mechanism(s) for the ion selectivity, we compare the power generation and transport rates of the VA-BNNT membranes for salts having different cation and anion diffusivities and thus diffusio-osmotic parameters.

5:58PM B36.00007 ABSTRACT WITHDRAWN

Saturday, November 23, 2019 4:40PM - 6:11PM —
Session B37 Turbulence: Particle Laden Flows

4:40PM B37.00001 Effect of fluid inertia on the orientation of a small spheroid settling in turbulence. BERNHARD MEHLIG, University of Gothenburg. We study the angular dynamics of small non-spherical particles settling in a turbulent flow. Most solid particles encountered in Nature are not spherical, and their orientations affect their settling speeds, as well as their collision and aggregation rates in suspensions. Whereas the random action of turbulent eddies favors an isotropic distribution of orientations, gravitational settling breaks the rotational symmetry. We demonstrate here that the fluid-inertia torque plays a dominant role in the problem. As a consequence rod-like particles tend to settle horizontally in turbulence, the more so the larger the settling number $S_v$ (a dimensionless measure of the settling speed). For large $S_v$ we determine the fluctuations around this preferential horizontal orientation for prolate particles with arbitrary aspect ratios, assuming small Stokes number $St$ (a dimensionless measure of particle inertia). This overdamped theory predicts that the orientation distribution is very narrow at large $S_v$, with a variance proportional to $S_v^{-1/4}$ for rods and $S_v^{-9/8}$ for disks. The abstract is based mainly on arXiv:1904.00481 (New Journal of Physics https://doi.org/10.1088/1367-2630/ab3062).

1Financial support by the grant Bottlenecks for particle growth in turbulent aerosols from the Knut and Alice Wallenberg Foundation, Dnr. KAW 2014.0048, VR grant no. 2017-3865, IDEXLYON project (Contract ANR-16-IDEX-0005) under University of Lyon auspices. Computational resources were provided by C2S2 and SNIC, and PSA2.

4:53PM B37.00002 Collision efficiency of rapidly settling particles in a turbulent flow. PIJUSH PATRA, ANUBHAB ROY, Department of Applied Mechanics, Indian Institute of Technology Madras, Chennai 600036, India, DONALD L. KOCH, Smith School of Chemical and Biomolecular Engineering, Cornell University, Ithaca, NY 14853, USA. We calculate the collision rate constant of hydrodynamically interacting low-sphericity spherical particle pairs sedimenting in a homogeneous isotropic turbulent flow in the rapid settling limit where the settling rate appears well described by the Kolmogorov eddy size. The result is expressed in terms of the velocity gradient auto-correlation function along the settling trajectory and in this particular problem due to rapid settling assumption we are able to relate it with the turbulence energy spectrum. A convection-diffusion equation for pair probability density function, $P(r,t)$, is derived in terms of the hydrodynamic turbulent pair diffusion and drift and then solved numerically to calculate the collision rate constant.
5:06PM B37.00003 Measurement of Inertial Particle Collision Statistics in Isotropic Turbulence Using 3D Particle Tracking Velocimetry¹. ADAM HAMMOND, ZACH LIANG, HUI MENG, University at Buffalo SUNY — We experimentally investigate the effects of turbulence and particle inertia on three particle collision statistics in particle-laden isotropic turbulence: radial distribution function (RDF), radial particle-pair relative velocity (RV), and for the first time, geometric collision kernel for which RDF and RV are factors. Experimentally obtaining these statistics has been difficult in the past especially as particle separation distance decreases below the Kolmogorov length scale $\eta$ to near-contact, where the physics becomes obscure and theoretical models may not hold. Using Shake-the-Box 3D particle tracking, we are able to resolve particle positions and velocities at $r \leq \eta$, which enables estimations of RDF and RV. Experiments are performed in a near-contact condition and compared to the colliion kernel. Experimental results are compared to previously calculated collision kernels for which RDF and RV are factors. RDF increases as $r/\eta$ decreases to 0.01, and increases with $St$. RV decreases as $r/\eta$ increases until $r/\eta = 1$, recapitulating previous DNS and experimental results (Dou et al., 2018, DOI: 10.1017/jfm.2017.813); however, as $r/\eta < 1$, RV exhibits a sharp upturn. This first simultaneous near-contact estimation of RDF, RV, and collision kernel calculated from real turbulence enables examination of theory and DNS, and may reveal new phenomena not previously accounted for in prior models.

¹This work was supported by NSF CBET-0967407, and CBET-1828544.

5:19PM B37.00004 Fall velocities of Hydrometeors in Turbulent Flow¹. AHMAD TALAEI, TIM GARRETT, University of Utah — An understanding of the interactions of precipitating aerosols, droplets and ice crystals within an atmospheric turbulent flow is fundamental to predictions of atmospheric weather and climate. Here we examine the mean settling velocity of a hydrometeor falling into a random Gaussian turbulent flow using hydrometeor images and velocities captured by the Multi-Angle Snowflake Camera (MASC) at Oliktok Point, Alaska. Analyses reveal hydrometeor Reynolds numbers ranging from 1 to 1000, sharply peaked at 200. Due to mathematical difficulties, previous analytical solutions of the equation of motion of a falling sphere in a viscous liquid have been constrained to the slowly falling particles in the Stokes regime with Reynolds numbers less than unity. In this study, we introduce an analytical solution for higher Reynolds numbers and develop an equation of motion for studying the interaction of atmospheric turbulence and hydrometers. The results show settling velocity reduction in weak turbulence and enhancement in strong turbulence.

¹The research is granted by Department of Energy Atmospheric System Research Award No. DE-SC001282

5:32PM B37.00005 Inertial particle velocity and distribution in vertical turbulent channel flow: a numerical and experimental comparison¹. DAVID RICHTER, GUIQUAN WANG, University of Notre Dame, KEE ONN FONG, FILIPPO COLETTI, University of Minnesota, JESSE CAPECELATRO, University of Michigan — This study is concerned with the statistics of vertical turbulent channel flow laden with inertial particles for two different volume concentrations ($\Phi_V = 3 \times 10^{-6}$ and $\Phi_V = 5 \times 10^{-5}$) at a Stokes number of $St = 58.6$ based on viscous units. Two independent direct numerical simulation (DNS) models utilizing the point-particle approach are compared to recent experimental measurements, where all relevant nondimensional parameters are directly matched. While both numerical models are built on the same general approach, details of the implementations are different. At low volume loading, both numerical models are in general agreement with the experimental measurements, with certain exceptions near the walls. At high loading, these discrepancies are increased, and it is found that particle clustering is overpredicted in the simulations as compared to the experimental observations. Potential reasons for the discrepancies are discussed. As this study is among the first to perform one-to-one comparisons of particle-laden flow statistics between numerical models and experiments, it suggests that continued efforts are required to reconcile differences between the observed behavior and numerical predictions.

¹ARO Grant G00003613-ArmyW911NF-17-0366 and ONR Grant N00014-16-1-2472

5:45PM B37.00006 Simultaneous tracking of suspended particles and time-resolved PIV in a turbulent boundary layer¹. FILIPPO COLETTI, LUCIA BAKER, University of Minnesota — A detailed picture of the interaction between suspended sediment and the carrier fluid has only recently begun to emerge due to recent advances in experimental and numerical methods. Here we investigate experimentally the dynamics of spherical particles in a turbulent boundary layer in a saltation-suspension transport regime. Particle image velocimetry and particle tracking velocimetry are used to obtain simultaneous, time-resolved fluid velocity fields and particle trajectories. Statistics of particle velocity, particle acceleration, and fluid velocity at particle locations are computed to characterize particle behavior and investigate mechanisms for particle deposition and resuspension. Fluid ejection events near the wall appear to be a main mechanism for particle suspension, while fluid sweeps contribute less to particle deposition. Particle acceleration variance is found to peak markedly near the wall, in response to passing turbulent structures. Resuspension is triggered by an increase in streamwise fluid velocity at particle location, while deposition is preceded by a decrease in fluid velocity. Resuspending particles experience much stronger wall-normal acceleration magnitude than depositing particles.

¹This work was funded by the Legislative-Citizen Commission on Minnesota Resources (LCCMR). L. J. Baker was supported by the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program.

5:58PM B37.00007 Shape- and scale-dependent coupling between inertialless spheroids and velocity gradients in turbulence. NIMISH PUJARA, University of Wisconsin - Madison, CRISTIAN LALESCU, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization — Particles of different shapes and sizes are commonly found suspended in a turbulent flow in the environment and in industrial processes. To better understand the dynamics of neutrally-buoyant, non-spherical particles in dilute concentrations, we compute the motion of inertialless spheroids in direct numerical simulations of turbulence using one-way forcing. Particles of different sizes are modelled as tracer bubbles after the velocity field has been coarse-grained at different filter scales. The focus is on the statistics of particle rotations and what they reveal about the interaction between the particles and velocity gradients. While particle rotations in the co-ordinate axes fixed to each particle show interesting variations with particle shape and filter scale, the mean-square value of particle angular velocity in the global co-ordinate axes is nearly constant across all shapes and scales. These trends are further probed by examining the particle alignment with fluid vorticity and how this depends on particle shape and filter scale. Finally, a comparison between these results and laboratory experiments provides insights into how particle inertia may influence particle-turbulence coupling for large, anisotropic particles.

Saturday, November 23, 2019 4:40PM - 6:11PM – Session B38 Porous Media Flow CO2 Sequestration and Dispersion 620 - Kenneth T. Christensen, University of Notre Dame
4:40PM B38.00001 CO₂ sequestration via pressure driven displacement on fluid-gas plugs in a capillary tube. SRAVYA SASETTY, THOMAS WARD, Iowa State University — This talk focuses on experiments conducted to further our understanding on the feasibility of carbon sequestration using a chemical reaction between CO₂ gas and aqueous Ca(OH)₂, which produces CaCO₃ precipitates. Experiments were performed in a capillary tube (dia ≈ 800 µm) by displacing liquid plugs of different volumes containing Ca(OH)₂ (0 ≤ c ≤ 20 mol m⁻³) dissolved in aqueous glycerol solution using CO₂ gas at pressures 0.2 psig ≤ P ≤ 1.0 psig. A CCD camera captured the displaced and displacing fluid interfaces and an in-house MATLAB code was used to measure both the mean Uₘ and tip Uₜ velocities. Subsequently, we measure the film thickness using the expression m = 1 − Uₘ/Uₜ where m is a measure of displaced fluid still remaining inside the tube. Surface tension values obtained using an in-house pendant drop tensiometer were used to calculate the capillary number Ca. We report the m versus Ca trends observed in our experiments and compare them against immiscible fluid displacement classical results.

4:53PM B38.00002 Resolving the pore-scale dynamics of multiphase flow of supercritical CO₂ and water in a 2D circular porous micromodel using high-speed microscopic PIV. YAOFA LI, GIANLUCA BLOIS, KENNETH CHRISTENSEN, University of Notre Dame — Multiphase flow of supercritical CO₂ and water in porous media is relevant to geologic carbon sequestration and enhanced oil recovery, among many other applications in the energy and environmental sectors. After nearly two decades of research, it is now apparent that many macroscopic flow behaviors are controlled by pore-scale physics down to the micrometer scale. Recent evidence suggested that transient flow events such as Haines jumps, occurring on the time scale of milliseconds, and the associated dynamic effects, can greatly influence the accuracy of predictive models if not accounted for. Moreover, wetting properties of the porous matrix pose a strong control on the observed physics and dynamics, thus challenging its microscopic and macroscopic descriptions. To this end, the pore-scale flow of water and CO₂ is quantified using high-speed micro-PIV under reservoir-relevant conditions in a 2D circular micromodel featuring randomly distributed pillars and variable wettability. Resolving the flow is enabled by the very high temporal and spatial resolutions of the measurements. Statistical analysis during both steady and transient flows is performed to gain further insight into the intermittent behaviors as well as to quantify the effects of wettability on such behaviors.

5:06PM B38.00003 Dispersive entrainment in gravity currents in layered porous media. CHUNENDRA K. SAHU, JEROME A. NEUFELD, University of Cambridge — We present experimental results to quantify mixing in gravity currents in layered porous media occurring due to dispersion between dense and light fluids. Dye-attenuation based laboratory experiments were performed in homogeneous medium and heterogeneous medium consisting of two or four layers of distinct permeabilities. These experimental results show that the effects of mixing are more prominent closer to the nose and less prevalent towards the source location. In layered media, the volume of entrainment is much higher than that in the homogeneous medium due to flow instabilities like over-riding and Rayleigh-Taylor instabilities, which result in higher mixing rates. These experimental results motivate a general mathematical model in which we exploit the large aspect ratio of these currents to formulate a depth-averaged model of the evolution of the mass and buoyancy. We assume that the entrainment of ambient fluid into the gravity current can be parameterised by the mean horizontal velocity and an entrainment coefficient. Based on our experimental measurements, we quantify the dependence of the dispersive entrainment on the number of layers and their permeabilities.

5:19PM B38.00004 Some effects of cross-bedding on tracer dispersion in porous formations. NEERAJA BHAMIDIPATI, ANDREW WOODS, University of Cambridge — The sequential deposition of sediment in fluvial settings often leads to structures known as cross-bedding, in which the geological strata are aligned at an angle to the horizontal and consist of alternating interbedded layers of fine and course sediment. This leads to anisotropy in the permeability, which tends to be substantially higher in the along-bed direction. Over time, many such layers build up leading to highly heterogeneous permeable rocks. Many such formations have relatively higher permeability and can be ideal candidates for carbon sequestration or may be involved in the subsurface hydrological system, and so there is considerable interest in the flow patterns which arise through such formations. We present a series of numerical calculations of the spreading of a pulse of tracer through such formations, based on the pattern of such heterogeneity from a number of rock outcrops. We analyse these observations and develop some low order models of the controls on the dispersion and spreading of a cloud of tracer as it moves downstream. Our models have relevance for interpretation of tracer tests for groundwater flows and also in interpreting patterns of flow during carbon sequestration.

5:32PM B38.00005 Dissolution-driven convection of partially miscible fluids in porous media¹. CHING-YAO CHEN, National Chiao Tung University, QIAN LI, WEI HUA CAI, Harbin Institute of Technology, ECKART MEIBURG, UC Santa Barbara — Dissolution-driven convection produced at a free interface between CO₂ and brine in porous media is numerically studied to mimic the complex diffusion and flow process in the geological sequestration of CO₂. A Darcy-Cahn-Hilliard model with a particular free energy distribution is employed to simulate the partially miscible feature between the CO₂ and the brine. Simulations reveal that the evolution of the CO₂ plume exhibits several distinct stages, such as triggering, growing, merging and damping. Consequently, for the temporal development of the solute flux we can distinguish three major time periods, such as free convection, constrained convection and shutdown. In the free convection period, the time-averaged solute flux decreases with Rayleigh number. On the other hand, the solute flux is independent of the Rayleigh number in the periods of constrained convection and shutdown. Correlations of the solute flux and the Rayleigh number are proposed based on the simulations, which are able to predict the solute flux and the dissolved quantity of CO₂ into brine.

¹Ministry of Science and Technology of Taiwan (MOST 105-2221-E-009-074-MY3), National Natural Science Foundation of China (Grant Nos. 51576051 and 51421063)

5:45PM B38.00006 Modeling solutal convection in porous media: from pore to Darcy scale. MARC HESSE, BAOLE WEN, Department of Geological Sciences, The University of Texas at Austin, SAHAR BAKHSHIAN, SEYYED HOSSEINI, Bureau of Economic Geology, The University of Texas at Austin — We model the transport process of convective carbon dioxide dissolution in porous media on both pore and Darcy scales. Numerical simulations of the Navier-Stokes equations are performed in two-dimensional granular packing with constant porosity. We vary the driving force by changing the grain/pore size and the density difference to explore the effect of mechanical dispersion on the convective pattern and flux. We upscale the problem and then perform the Darcy-scale simulations with Fickian mechanical dispersion. The Darcy model reproduces both the trend in the convective pattern and quantitative fluxes of the pore-scale results, via adjusting the longitudinal dispersivity and the anisotropy of mechanical dispersion. Our simulations show the flux recovers a linear scaling with reduced coefficient as dispersion becomes dominant, consistent with the recent laboratory experiments. However, sub-linear flux scaling arises either in a transitional regime where diffusion and dispersion are comparable or if grains become too coarse. In this case, the pore-scale simulations show that convective up- and downwellings arise in two-dual pores. This leads to additional mixing not accounted for in Fickian dispersion model on the Darcy scale. The buoyancy-driven pore-scale mixing observed elsewhere has different characteristics from mixing processes in pressure-driven flows and requires a new approach to upscale them to the Darcy scale.
5:58PM B38.00007 The effect of non-uniform poroelastic reservoir geometry on the propagation of a buoyant gravity current. ADAM BUTLER, ALEX COPLEY, JEROME NEUFELD, University of Cambridge — Gravity currents in confined porous media occur in a variety of geological settings, with CO$_2$ storage one of particular current importance. How the CO$_2$ current and the porous medium develop over time is sensitive in particular to the pressure in the ambient fluid. This is set by compressibility and poroelastic deformation of the porous medium, and in this manner the current may experience the complex geometry of the domain. Here we model the flow of a gravity current injected into such a porous medium, exploiting the aspect ratio of a typical aquifer in order to vertically average the flow and medium properties across the domain. We derive coupled advection-diffusion equations for the injected and ambient phases and focus on the ambient phase in the limit of large Young’s modulus. Using a simple elastic-layer model, we incorporate the response to deformation from the overburden and calculate the diffusion of pore pressure away from the injection point as well as the resulting deformation transmitted to the surface. As a specific case study, we apply our model to an aquifer with converging lid and basement in order to investigate the effect of non-uniform reservoir geometry on propagation.

Saturday, November 23, 2019 6:30PM - 8:00PM — Session B39 Tutorial for Authors and Referees (6:30pm - 8:00pm) Sheraton Grand Hotel Grand Ballroom D -

6:30PM B39.00001 Tutorial for Authors and Referees (6:30pm - 8:00pm) —

Saturday, November 23, 2019 8:00PM - 10:00PM — Session B40 Meet the Physical Review Journal Editors Reception (8:00pm - 10:00pm) Sheraton Grand Hotel Grand Ballroom D -

8:00PM B40.00001 Meet the Physical Review Journal Editors Reception (8:00pm - 10:00pm) —

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C01 Turbulent and Chemically-reacting Flow Modeling I 2A - Peyman Givi, University of Pittsburgh

8:00AM C01.00001 Edward E. O’Brien’s Seminal Contributions to Turbulence Theory, FOLUSO LADEINDE, Stony Brook University, CESAR DOPAZO, Universidad Zaragoza, PEYMAN GIVI, University of Pittsburgh — A brief overview will be presented of the influential contributions of Edward E. (Ted) O’Brien to the theory of turbulence, with an emphasis on scalar mixing and reaction. While perhaps best known for his work on the transported PDF methods, Ted’s contributions are very diverse and consider a broad range of theoretical and computational approaches. In the 1960s, he made some very fundamental contributions to the spectral theory of reactive scalars, analyzed the consequences of passive scalar tagging using Corsin’s “backward Lagrangian diffusion” concept, and contributed to the interpretation of Kraichnan’s “direct interaction approximation” (DIA) for turbulent mixing. In the 1970s-1980s, he focused on scalar PDF Functional and Function methods. In fact, he is widely recognized for introducing and popularizing single- and multi-point PDF closures, as well as the scalar-gradient PDF within the reactive turbulent flow community. In the 1990s, he focused on applying the EDM and the “amplitude mapping closure” (AMC) models, respectively to reactive turbulent scalars and mixing. With wider availability of supercomputers in the late 1990’s-2000’s, Ted utilized DNS for the development and appraisal of modern turbulence closures. He is also credited with introducing the “filtered density function” (FDF) transport equation for LES of turbulent reactive flows. In fact, he is the first to develop a transported scalar-FDF equation for multi-species turbulent reactive flows. Professor O’Brien’s publications continue to be highly cited within the turbulence research community.

8:13AM C01.00002 Numerical Simulation of Colorless Distributed Combustion with LES/FMDF, HUSAM ABDULRAHMAN, FARHAD JABERI, Michigan State University, ABDOULAHAD VALIDI, ANSYS Inc., ASHWANI GUPTA, University Of Maryland — Honoring Ted O’Brien. Turbulent mixing and combustion in non-premixed and premixed Colorless Distributed Combustion (CDC) systems are studied with the hybrid large eddy simulation/filtered mass density function (LES/FMDF) methodology and its Eulerian–Lagrangian computational solver. The CDC has shown to significantly reduce NOx and hydrocarbon emissions while improving the reaction pattern factor and stability with low pressure drop and noise. The combustion in CDC is distributed and is characterized by wide fluctuations in flow variables. In addition to non-conventional distributed turbulent reaction, mixing between fuel, oxidizer, and combustion products in CDC is unique and complex. The LES/FMDF model is shown to be able to capture all the unique features of turbulent mixing and combustion in CDC. The consistency of the Eulerian and Lagrangian parts of LES/FMDF is established for both non-reacting and reacting conditions. The LES/FMDF results are shown to be in good agreement with the available experimental data. The numerical results indicate that the variations in the inflow air temperature, jet velocity and composition and premixing have a significant effect on the flow, mixing and combustion in the CDC. They also indicate the importance of jets setup in the combustor.

8:26AM C01.00003 Filtered Mass Density Function for Large-Eddy Simulations of Multiphase Turbulent Reacting Flows, FARHAD JABERI, Michigan State University, ZHAORUI LI, Texas A&M University-Corpus Christi, ARAZ BANAEEZADEH, Altair Engineering Inc., ABOLFZAL IRANNEJAD, Alcon-Novartis Inc. — Honoring Ted O’Brien. The filtered mass density function (FMDF) methodology is further extended and employed for large-eddy simulations (LES) of multiphase turbulent reacting flows. The two-phase LES/FMDF model is implemented with a unique Lagrangian-Eulerian-Lagrangian mathematical/computational methodology. In this methodology, the filtered carrier gas velocity field is obtained by solving the filtered form of the compressible Navier-Stokes equations while the gas scalar (e.g., temperature and species mass fractions) field and the liquid (spray) phase are obtained by solving two different sets of Lagrangian equations. The two-way interactions between the phases and all the Eulerian and Lagrangian fields are included in the two-phase LES/FMDF methodology. The accuracy and reliability of the model is demonstrated by comparing the two-phase LES/FMDF results with those obtained from the direct numerical simulation (DNS) and experimental data for a range of fundamental and practical multiphase flows including a spatially developing turbulent mixing layer with evaporating and reacting droplets and spray combustion in a preheated high-pressure closed chamber, a dump combustor, a double-swirl burner, and an internal combustion engine.
8:39AM C01.00004 A High-Order FDF Large Eddy Simulator of Complex Flows .
SHERVIN SAMMAK, Center for Research Computing, University of Pittsburgh, AIDYN AÎTZHAN, ARASH NOURI, PEYMAN GIVI, Department of Mechanical Engineering and Materials Science, University of Pittsburgh — Honoring Ted O’Brien. The flow solvers in most previous LES-FDF are based either on high-order discretization schemes in simple flows, or low-order (finite-volume) methods in complex flows. In this work, we develop a new computational methodology which allows LES of complex flows via the use of a high-order spectral-hp element scheme. The high order accuracy of the spectral discretization and the versatility of the finite element domain decomposition, facilitate high-fidelity simulation of flows within complex geometries. This CFD solver is combined with a Lagrangian Monte Carlo scheme for LES of a bluff-body reacting flow via the FDF subgrid scale closure [1]. Demonstrations are made of the consistency and the overall superior performance of this high order hybrid scheme. [1] Gao, F. and O’Brien, E. E., “A Large-Eddy Simulation Scheme for Turbulent Reacting Flows,” Phys. Fluids A, vol. 5(6), pp. 1282-1284 (1993).

8:52AM C01.00005 On Large Eddy Simulation/Filtered Density Function based Modeling of Circular Bluff Body Configurations . CESAR CELIS, RICARDO FRANCO, Pontificia Universidad Catolica del Peru (PUCP) — Honoring Ted O’Brien. Large eddy simulation/filtered density function (LES/FDF) numerical results of inert and reacting flows characterizing the near wake of bluff body configurations are discussed in this work. Circular bluff body configurations are studied because they feature strong interactions between turbulence and chemical reaction, as well as pollutants formation. All numerical results obtained here are compared to experimental data gathered previously. Parameters particularly analyzed include velocity profiles, turbulent kinetic energy, Reynolds stress and strain rate tensors. A strong anisotropic flow is observed from the obtained results along with a flow recirculation zone consisting of a toroidal vortex. At inert conditions, large turbulent fluctuations are found at the stagnation point region. The observed flow anisotropy is associated with the stagnation point flow. The results discussed here correspond to on-going work involving both bluff body burner configurations and numerical predictions of rather complex phenomena such as soot formation.

9:05AM C01.00006 Molecular mixing in highly turbulent premixed flames1, XINYU ZHAO, PATRICK MEACHER, University of Connecticut — Honoring Ted OBrien. The molecular mixing rules and rates in premixed flames subject to intense turbulence are investigated in this study. Direct numerical simulation (DNS) of a spherical product kernel is conducted in a homogeneous isotropic turbulence box. The triply periodic computational domain outside the product kernel is comprised of fresh mixtures. The transient flame kernel undergoes flame propagation, local extinction, and eventually global extinction. During the transition, the compositional space evolves from a low-dimensional manifold to increasingly higher dimensions. The DNS data are subsequently explicitly filtered to study the subgrid-scale behavior of the scalars. The Euclidean minimum spanning trees are constructed to understand the change of localness during the extinction process. Conditional statistics of major and minor species are collected, according to the mixing rules of various mixing models. A scalar gradient based mixing frequency model is constructed and assessed for its suitability to represent the mixing rates of critical species during all phases of the flame kernel evolution.

9:18AM C01.00007 Uniform mean scalar gradient in grid turbulence: Asymptotic probability distribution of a passive scalar , XIAODAN CAI, United Technologies Research Center — Honoring Ted O’Brien. Dr. Edward E. O’Brien was my Ph.D advisor in mechanical engineering at Stony Brook University. It was he who introduced me to study flow turbulence in the United States after we met at a Fluid Dynamics conference in Beijing. He was an extremely humble, patient and optimistic person, and was an inspiration to all. Dr. O’Brien stressed the importance of understanding the fundamentals and was rigorous in applying them to solve important problems. I am one of Professor O’Brien’s students who has benifited immensely from his approaches and values. I will now present our work on asymptotic behaviors of probability distribution for a passive scale in grid turbulence, which highlights Professor O’Brien’s legacy.

9:31AM C01.00008 Modeling Radiative Heat Transfer and Turbulence-Radiation Interactions Using PDF and FDF Methods , DANIEL HAWORTH, The Pennsylvania State University — Honoring Ted OBrien. In 1974, Dopazo and O'Brien proposed using a modeled equation for the probability density function of a set of scalar variables that describe the thermochemical state of a reacting medium (a transported composition joint PDF) to model mixing and reaction in chemically reacting turbulent flows. Since then, the benefits of PDF methods, including subsequent extension to large-eddy simulations (filtered density function (FDF) methods), for modeling turbulence-chemistry interactions have been well established. Those benefits are a consequence of the ability of PDF/FDF methods to represent the influences of unresolved turbulent fluctuations on one-point physical processes (such as chemical reactions) in a natural way. For the same reason, PDF/FDF methods have an advantage in dealing with the influences of unresolved turbulent fluctuations on radiative emission. And when coupled with a stochastic radiation solver, the benefits can be extended to radiative absorption, thereby capturing both emission and absorption turbulence-radiation interactions. A model that combines stochastic Lagrangian particle PDF/FDF methods and a photon Monte Carlo method for radiative transfer is presented. Results are presented for laboratory flames and high-pressure combustion systems.

Presentations:

**Investigation of scalar-scalar-gradient filtered joint density function for large eddy simulation of turbulent combustion**

9:57AM C01.00010 Investigation of scalar-scalar-gradient filtered joint density function for large eddy simulation of turbulent combustion

CHENNING TONG, Clemson University — Honoring Ted O’Brien. The scalar-scalar-gradient filtered joint density function (FJDF) is studied experimentally. Measurements are made in the fully developed region of an axisymmetric turbulent jet (with a jet Reynolds number of 40000) using an array consisting of three X-wires and three resistance-wire temperature probes. Filtering in the cross-stream and streamwise directions are realized by using the array and by invoking Taylor’s hypothesis, respectively. The measured mean FJDF conditional on the (subgrid-scale) SGS scalar variance is unimodal when the SGS scalar variance is small compared to its mean. The scalar gradient depends weakly on the SGS scalar. For large SGS variance the FJDF is bimodal and the gradient depends strongly on the SGS scalar. The SGS scalar under such a condition contains diffusion layer structures and the SGS mixing is similar to the early stages of binary mixing. The iso-scalar surface in the diffusion layer has a lower surface-to-volume ratio than those in a well mixed scalar. The conditionally filtered diffusion of the scalar gradient has a S-shaped dependence on the scalar gradient, which is expected to be qualitatively different from that of a reacting scalar under fast chemistry conditions. However, because modeling is performed at a higher level and because the scalar-scalar-gradient FJDF contains the information about the scalar dissipation and the surface-to-volume ratio, the FJDF approach is expected to be more accurate than scalar filtered density function approaches and has the potential to model turbulent combustion over a wide range of Damkohler numbers.

**Supported by NSF**

---

**Progress and Challenges for Computational Naval Hydrodynamics**

8:00AM C02.00001 Physics of Naval Flows

THOMAS FU, Office of Naval Research — The flow past a ship operating in the ocean is one of our most challenging physics problems. Looking at a ship operating in waves, we see a fluid sheet separating off the bow, a multiphase contact line along the hull waterline, spray and bubbles from wave breaking, air wakes, and Kelvin ship waves. Below the waterline, we see turbulent boundary layers, D-3 flow separation and vortices, cavitation, and the rotational flow from the propeller. We also have the ocean itself adding a layer of complexity. A marine platform interacts with the world through environmental and hydrodynamic forcing. The environment (waves, wind, and buoyancy/gravity) acts on the platform, and it acts on the environment (propulsor, hull, and control surfaces). However, a naval platform’s performance is characterized by more than just its performance as a ship, and in fact, it might be a submarine, planning craft, or amphibious vehicle, and the performance is also now characterized by its ability to perform military missions and survive. To the list of technical areas we have already mentioned, fluid dynamics and oceanography, we can now add acoustics, heat transfer, and meteorology. As we have noted the complexity of naval flows due to the range of physical phenomena (turbulence, waves, wave breaking, cavitation, boundary layers, etc.), we should also note the complexity due to the range of spatial and temporal time scales involved. For the phenomena already mentioned, we are interested in length scales ranging from Kolmogorov to Rossby.

**Office of Naval Research**

---

**Physics of Surface Piercing Bodies**

8:52AM C02.00003 Physics of Surface Piercing Bodies

ANNE FULLERTON, Naval Surface Warfare Center — Most Navy vehicles operate at or near the free surface in order to meet mission requirements. As such, it is important to be able to accurately model and understand the physics of surface piercing bodies to inform the Navy of the vehicle’s performance during these operations. Continual advances in computing power and the ever-increasing availability of supercomputing resources have changed the way computational physics solvers are used in the Navy. However, even with modern High Performance Computing (HPC) resources, it is not currently feasible to directly simulate all of the relevant physics of Navy vehicles beneath, at, and above the free surface for necessary real-world time scales. Most modern computational physics codes address these shortcomings by resolving down to the smallest scales allowed by given computational resources and modeling phenomena that occur below the finest resolved scales. This work will discuss necessary physics of surface piercing bodies, current modeling techniques, and how they ultimately affect predictions which are used to inform the fleet.
High Reynolds Number Turbulent Flow will reduce skin friction drag at high Reynolds numbers (e.g., micro-fluidic devices). Yet, SHSs influence on fully developed turbulent boundary layers (TBL) is unclear. In order to develop SHSs that are related to the physics, material science, and manufacture of a wide variety of SHSs. Liquids undergoing laminar flow over SHSs may develop a Cassie-Baxter state at the flow boundary, and the presence of the gas pockets between rough surface asperities leads to a reduction in the shear stress compared to that of a smooth solid boundary, especially in flow channels where the slip length is on the order of the channel dimension (e.g., micro-fluidic devices). Yet, SHSs influence on fully developed turbulent boundary layers (TBL) is unclear. In order to develop SHSs that will reduce skin friction drag at high Reynolds numbers ($Re \geq 10^5$), our multi-university research group conducted a five-year effort focusing on five areas: (1) creation of optimized SHSs for high Reynolds number – we developed methodologies to manufacture specific SHSs with the desired surface energy and topology for the experimental examination of TBL modification at high Reynolds numbers; (2) examination of the interaction of near-wall flows with micro-scale surface gas pockets; (3) examination of the turbulence modification of micro-structured SHSs on five areas; (4) development of SHSs for passive and active gas replenishment to develop and maintain the Cassie-Baxter condition; and (5) experimental validation of SHS drag reduction at high Reynolds Numbers. This talk will present an overall summary of the efforts results and conclusions.

---

Sunday, November 24, 2019 8:00AM - 10:10AM – Session C03 Minisymposium: Fluid Dynamics in the Clinical Management of Intracranial Aneurysm 201 - Michael Levitt, University of Washington

8:00AM C03.00001 The use of fluid dynamics to predict success of intracranial aneurysm endovascular treatment: State of the Art and Potential for Translation to Clinical Use, M. LEVITT, M. BARBOUR, L. MARSH, V. CHIVUKULA, C. CHASSAGNE, C. KELLY, S. LEVY, Y. ZHENG, L. KIM, A. ALISEDA, University of Washington — Endovascular placement of coils or stents have become the preferred method for treatment of intracranial aneurysms. These minimally invasive procedures aim to create hemodynamics conditions that promote a stable thrombus that fills the entire aneurysm sac. No accurate exists, with failure to create a sac-filling thrombus occurring in 15-35% endovascular treatments that need retreatment involving significant risk to the patient and cost to the health care system. We simulate 60 patients treated with Flow Diverting Stents (FDS) or coils, both before and immediately after treatment. Synchrotron scans of the actual coils or stents deployed inside phantoms of the aneurysms are generated, allowing for comparison of the hemodynamics modeled with standard porous media versus homogenized anisotropic and heterogeneous models derived from the fully-resolved scans. We have developed both Eulerian and Lagrangian metrics to understand the hemodynamics that promote a successful embolization of the aneurysmal sac, and propose a risk score to predict the need to further treat or follow-up each patient during, or immediately after, endovascular treatment.

8:26AM C03.00002 Modeling of patient-specific blood flow in the brain: Are we there yet?, KRISTIAN VALEN-SENDSTAD, Simula Research Laboratory — A brain aneurysm is a focal weakening of the arterial wall, and rupture is associated with high rates of morbidity and mortality. Roughly

8:52AM C03.00003 Improving clinical decision-making with data: defining roles for computational fluid dynamics in aneurysm management, JASON M. DAVIES, Dept. of Neurosurgery & Dept. of Biomedical Informatics, University at Buffalo — Subjectivity plays an outsized role in the management of intracranial aneurysms (IA). Computational fluid dynamics may reduce variability and improve outcomes. In this study, we aimed to explore the utility of CFD in the management of UIAs in two domains, 1) identification of aneurysms that should be treated, and 2) prediction of occlusion outcomes for flow-diverted-treated aneurysms.

Methods: We identified aneurysm patients between September 2018 and June 2019 and computed the unruptured intracranial aneurysm treatment score (UIATS) as well as the Rupture Resemblance Score (RRS). We further identified patients treated with flow-diveters between 2011 and 2018. We computed 6 geometrical ($G$), 2 FD device-related ($D$), 4 pre-treatment (untreated, $U^*$) and 4 post-treatment (treated, $T^*$) hemodynamic features, which were then used in combination to train four neural network models.

Results: Treatment decisions were analyzed for 49 patients with 84 UIAs. Of the 42Conclusions: Hemodynamics contribute information relevant to the management of intracranial aneurysms. Incorporating data-driven management strategies should reduce variability in outcomes and improve patient care.

In collaboration with: Nikhil Paliwal, Dept. of Mechanical & Aerospace Eng. & Canon Stroke and Vascular Research Center, Univ. of Buffalo; Hui Meng, Dept. of Mechanical & Aerospace Engineering, Canon Stroke and Vascular Research Center, Dept. of Biomedical Engineering, Dept. of Neurosurgery, Univ. of Buffalo.

1. This project is funded by ONR grant number: N00014-16-1-3188.
2. Funded by the U.S. Office of Naval Research (ONR) MURI (Multidisciplinary University Research Initiatives) program (Grant No. N00014-12-1-0874) managed by Dr. Ki-Han Kim.
9:18AM C03.00004 Hemodynamics and Aneurysm Formation. MIN PARK, University of Virginia Health Center — Aneurysms develop due to a multitude of factors. Traditionally, research surrounding aneurysm formation has examined an individual's lifestyle and medical co-morbidities, such as hypertension, nicotine use, and a pre-existing family history. With improvements in technology and the ability to more accurately model hemodynamic forces, research has also focused on the role of hemodynamic forces on aneurysm development, remodeling, and the risk of aneurysm rupture. Computational fluid dynamic studies allow researchers to quantify the effects of fluid movement within a vessel and aneurysm. They can also identify parameters, such as wall shear stress and oscillatory shear index, which likely induce changes within the vessel to promote aneurysm formation and remodeling through complex biological pathways. We will review the current knowledge and beliefs regarding aneurysm formation and examine how computational fluid dynamics has added to our understanding of aneurysm pathophysiology and how it influences clinical decision making.

9:44AM C03.00005 Computational Fluid Dynamics and aneurysm wall mechanobiology: correlations between CFD, MRI vessel wall imaging and histology. MAHMOUD MOSSA-BASHA, University of Washington — TBD

Sunday, November 24, 2019 8:00AM - 10:10AM
Session C05 Sprays/Droplet Combustion and Ignition 204 - Jacqueline H. Chen, Sandia National Laboratories

8:00AM C05.00001 Flame structure analysis and flame stabilization in a turbulent swirling spray flame\(^1\), DANYAL MOHADDES, WENWEN XIE, MATTHIAS IHME, Stanford University — The quantitative prediction of the flow and combustion dynamics within highly turbulent environments encountered in modern aviation gas turbine engines remains an important challenge for the numerical combustion community. The use of large eddy simulation (LES) has become well-established for the analysis of such flows. Although a multitude of modelling approaches exist for combustion chemistry with known limitations in accuracy and computational cost, specific effects of a combustion model on a given simulation cannot be known a-priori. In this study, a turbulent swirling n-dodecane spray flame at ambient pressure is investigated using LES employing the Lagrangian point-particle approach and the eddy-viscosity model. The quantitative prediction of the flow and combustion dynamics within highly turbulent environments is studied. Experiments in the ZARM 4.7 s drop tower are conducted using a cryogenically-cooled test chamber that allows for pressures up to 60 bar. Initial experiments indicate that the hydrogen-oxygen diffusion flame is formed relatively close to the droplet surface. During the combustion process the surface of the LOX droplet appears to become covered by a water-ice layer, which ruptures to produce discrete, gaseous oxygen jets. External to the flame zone, the water vapor combustion product is observed to condense or freeze, forming a spherical shell around the burning droplet. The flame standoff distance and the droplet regression rate are investigated with shadowgraphy and OH chemiluminescence imaging. The experimental results are compared with the findings of numerical simulations conducted by the University of Washington.

8:13AM C05.00002 Liquid Oxygen Droplet Combustion in Hydrogen under Microgravity Conditions\(^1\), FLORIAN MEYER, CHRISTIAN EIGENBROD, ZARM Center of Applied Space Technology and Microgravity, VOLKER WAGNER, WOLFGANG PAA, IPHT Leibniz Institute of Photonic Technology, JAMES HALL, MICHAEL ZODY, JON FRYDMAN, JAMES HERMANSON, University of Washington — In liquid rocket propulsion the liquid oxygen (LOX) and liquid hydrogen system is widely used. Single oxygen droplets burning in gaseous hydrogen surrounds are investigated, representing the most basic element of this spray combustion process. The basic processes of droplet vaporization, mixture formation, ignition and combustion under cryogenic conditions in microgravity are studied. Experiments in the ZARM 4.7 s drop tower are conducted using a cryogenically-cooled test chamber that allows for pressures up to 60 bar. Initial experiments indicate that the hydrogen-oxygen diffusion flame is formed relatively close to the droplet surface. During the combustion process the surface of the LOX droplet appears to become covered by a water-ice layer, which ruptures to produce discrete, gaseous oxygen jets. External to the flame zone, the water vapor combustion product is observed to condense or freeze, forming a spherical shell around the burning droplet. The flame standoff distance and the droplet regression rate are investigated with shadowgraphy and OH chemiluminescence imaging. The experimental results are compared with the findings of numerical simulations conducted by the University of Washington.

8:26AM C05.00003 Numerical Simulation of Liquid Oxygen Droplet Combustion in Hydrogen\(^1\), JAMES HALL, MICHAEL ZODY, JON FRYDMAN, JAMES HERMANSON, University of Washington, FLORIAN MEYER, CHRISTIAN EIGENBROD, ZARM Center of Applied Space Technology and Microgravity, VOLKER WAGNER, WOLFGANG PAA, IPHT Leibniz Institute of Photonic Technology — In liquid rocket propulsion oxygen normally enters the combustion chamber as a dispersed phase, while the hydrogen fuel rapidly evaporates into a continuous, vapor phase. The ignition and combustion of a single, liquid oxygen droplet in gaseous hydrogen surroundings is thus the essential, first step in the subsequent spray combustion process. The OpenFOAM platform is used to calculate species concentrations, temperatures, heat release, and reactant consumption for this system. The simulations suggest that ignition, subsequent to the initial diffusion of gaseous oxygen into hydrogen, initially results in the appearance of two flame zones. As quasi-steady combustion is approached, these two flames merge. The resulting stable, quasi-steady flame, combined with the conductive heat transfer from the flame to the droplet surface is used to predict the oxygen droplet-combustion lifetime. These numerical simulations are conducted in parallel with drop-tower tests at ZARM. The calculated lifetime of a 1-mm liquid oxygen droplet undergoing combustion in gaseous hydrogen at 1 bar pressure and an initial temperature of 100 K is comparable to that observed in the drop-tower tests.

8:39AM C05.00004 ABSTRACT WITHDRAWN —

8:52AM C05.00005 Ignition Kernel Dynamics in a \(M = 3\) Flame holder. ESTEBAN CISNEROS-GARIBAY, University of Illinois at Urbana-Champaign, DAVID BUCHTA, The Center for Exascale Simulation of Plasma-Coupled Combustion, University of Illinois at Urbana-Champaign, JONATHAN FREUND, University of Illinois at Urbana-Champaign — The coupled mixing and reaction time scales of ignition in a supersonic flame-holding cavity flow are studied with detailed numerical simulations. A round jet ejects ethylene into the cavity under a \(M = 3, T = 440 \) K free-stream. The ignition (and subsequent sustained flame) are studied in detail, including direct comparisons with corresponding measurements. Two injection configurations are simulated: (i) vertical, from the cavity floor; and (ii) horizontal, from the cavity's 45° back wall. Ignition is seeded by a laser-induced breakdown (LIB), which creates radical species and locally heats the gas. Comparisons are made against measured excited hydroxyl radical (OH*) to assess prediction accuracy. The injection geometry significantly affects the direction in which ignition kernels (as quantified by OH mass fraction) advect and grow. Over the first 75 \(\mu s\), the turbulence mechanics that produce this and the observed large variance of ignition kernel statistics are studied and compared. These variances are greater than dependencies on the chemical kinetic parameters.
Temperature Compression Ignition Environments

9:05AM C05.00006 Direct Numerical Simulation of Multi-Injection Ignition in Low-Temperature Compression Ignition Environments
MARTIN RIETH, Sandia National Laboratories, MARC DAY, Lawrence Berkeley National Laboratory, SHUBHANGI BANSUDE, TIANFENG LIU, University of Connecticut, CHOL-BUM KWEO, JACOB TEMME, Army Research Laboratory, JACQUELINE CHEN, Sandia National Laboratories — Multi-injection strategies in combustion engines are known to be able to reduce pollutant emissions and improve ignition reliability. The latter is especially important in advanced compression ignition engine concepts relying on low-temperature high-efficiency operation or for engines operating at extreme high-altitude conditions. Under low-temperature conditions, fluid from the first injection does not ignite prior to mixing with fluid from the second injection, but provides pre-ignition chemical species and potentially a moderately elevated temperature. Fluid from the second injection mixes with this partially reacted mixture and its ignition is accelerated. However, the exact details of how pre-ignition species accelerate the ignition of the fluid from the second injection are not known. We will explore this with Direct Numerical Simulations (DNS) and, in particular, we will focus on understanding how particular chemical species and reaction pathways are responsible for accelerated ignition, identify the combustion modes and flame topologies present during ignition, and how these are affected by turbulent mixing and entrainment. To this end, we utilize a chemical explosive mode analysis, reaction-diffusion balances and related techniques.

9:18AM C05.00007 The impact of ignition on the occurrence and dynamics of multi-stage flames under shock tube conditions
TIANHAN ZHANG, YIGUANG JU, Princeton University — The laminar flame speeds and structures of ignition-assisted cool and warm n-heptane/air flames are studied computationally and analytically. The primary objective is to understand the effects of the ignition Damkohler number, mixture temperature, equivalence ratio, and pressure on the dynamics and structures of cool and warm flame propagation under shock tube conditions. Different transitions among cool, warm, and hot flames are examined. The results show that both cool and warm flame structures and propagation speeds change dramatically with the increase of the ignition Damkohler number and are affected by the initial temperature, equivalence ratio, pressure, and flame regimes. Furthermore, within the hot flame flammability limits, it shows that the cool flame speed has a non-monotonic dependence on the initial mixture temperature due to the negative temperature coefficient (NTC) effect, while the hot flame speed is divergently different and only increases monotonically with the initial temperature. Finally, compared with recent flame speed measurements using the shock tube, the simulation agrees well with the experimental data and demonstrates clearly how high speed and multi-stage flames affect the experimental observation and result in the NTC behavior.

9:31AM C05.00008 Three-dimensional effects in vorticity production, cellular instabilities, and transition to turbulence in focused-laser-induced ignition kernels
JONATHAN F. MACART, JONATHAN M. WANG, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — Ignition of combustible mixtures via laser-induced breakdown (LIB) involves interactions between thermal, chemical, and hydrodynamic processes. An initially axisymmetric plasma core has large density, pressure, and velocity gradients that lead to vorticity production, collapse of the plasma core, and a transition to three-dimensional evolution. These stages and their co-evolution with the thermochemical state are investigated in lean hydrogen-oxygen premixtures using three-dimensional detailed numerical simulations, in which the compressible Navier-Stokes and reactive species equations are solved assuming local thermodynamic equilibrium and charge neutrality. The initial LIB and gasdynamics are simulated in auxiliary calculations employing a two-temperature non-local thermodynamic equilibrium kinetic model and a radiative transfer equation. Three-dimensional vorticity is observed in the plasma core, but this initial vorticity diffuses during the collapse of the core and transition to laminar burning. After an initial period of laminar propagation, the flame develops a cellular instability, which accelerates the flame-front and produces flame-front vorticity leading to sustained turbulence.

9:44AM C05.00009 Hydrodynamic Ejection from a Laser-induced Breakdown and its Implications for Ignition
JONATHAN WANG, University of Illinois at Urbana-Champaign, DAVID BUCHTA, JONATHAN MACART, The Center for Exascale Simulation of Plasma-coupled Combustion, University of Illinois at Urbana-Champaign, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Optical breakdown of a gas by a focused laser produces a high-temperature, high-pressure plasma kernel that expands rapidly and, in some cases, ejects hot gas along the laser axis. Traveling as a hot vortex ring, this ejected gas can reach distances several times the size of the plasma kernel and, in a combustible mixture, initiate flame growth away from the laser focal region. Under certain conditions (e.g. sub-atmospheric pressure), however, the ejection can fail to form or even reverse direction, altering the distribution of heat and radical species necessary for ignition. Detailed simulations of a model kernel, confirmed to reproducibly key experimental observations, show how the ejection and its reversal are caused by two primary mechanisms of vorticity production: tangential variations in the strength of the shock produced by the breakdown, and baroclinic generation in the trailing rarefaction. Relatively mild changes in the early-time kernel geometry alone—including, for example, a 20% increase in overall aspect ratio—can alter the subsequent interaction of auto-advecting vorticity and ultimately precipitate ejection failure or reversal. The ejection influences ignition via the competing effects of strain and heating.

9:57AM C05.00010 Thermal-Chemical Instability in Plasma-Assisted Combustion
HONGTAO ZHONG, MIKHAIL SHNEIDER, Department of Mechanical and Aerospace Engineering, Princeton University — In the flow of weakly ionized plasma, the transition from a diffusive volumetric discharge to a contracted localized filament is called plasma thermal instability. Traditionally, plasma thermal instability is controlled by the well-known thermal-ionization mechanism. However, endothermic/exothermic chemical reactions may bring new couplings between the reactive flow of weakly ionized plasma and chemical kinetics. In this work we developed a one-dimensional numerical model and studied the dynamics of thermal-chemical instability in a reactive mixture. Several key parameters including flow speed, gas pressure, initial temperature, and mixture composition are examined for their influence on triggering the thermal-chemical instability. This work will fill the knowledge gap in understanding the chemical kinetic effect on plasma instability in combustible mixtures and provide support for the future development of volumetric plasma ignition for ultra-lean combustion in advanced engines.

Sponsors: DOE ECP/DOE VTO/DoD Army

This work was supported by NSF and DOE Grant on plasma instability and plasma chemistry.
Details of its structure were unknown in part due to difficulties in experimental diagnostics. This work presents numerical simulations of the blue whirl, which reveals its flow structure and burning modes for the first time. The numerical model solves the unsteady, compressible, Navier-Stokes equations. The effects of combustion are modeled using a calibrated chemical-diffusive model. The domain is a cubical enclosure with sides which are 30 cm long. Circulation is imposed by forcing air through the corners and a constant flux of fuel is specified at the center of the bottom floor within a specified diameter. The upper boundary is an outflow condition and all other boundaries are non-slip. The results reveal a triple flame structure within the blue whirl and the influence of vortex breakdown on its burning and flow structure.

1 National Science Foundation, Army Research Office

8:13AM C06.00002 Numerical description of blue whirls over liquid-fuel pools1. ANTONIO L SANCHEZ, University of California San Diego, JAIME CARPIO, Universidad Politecnica de Madrid, WILFRIED COENEN, University of California San Diego, ELAINE S. ORAN, Texas A&M University — Previous experiments of liquid-pool fires with ambient swirl have shown that, when the outer circulation is increased beyond a critical value, the existing fire whirl transitions to a new, fundamentally different configuration involving a lifted partially premixed front lying below a recirculating-flow bubble. The steady axisymmetric structure of this so-called blue whirl is investigated here by numerical methods. The problem is formulated with account taken of the boundary velocity induced by the thermal plume developing vertically above the axis, with a one-step Arrhenius description adopted for the heat release and a simple, optically thin approximation for H2O and CO2 radiation, the latter being fundamental for fuel vaporization. Integrations for increasing values of the swirl level are seen to describe all of the flow features observed in experiments, including the detachment and subsequent lifting of the flame edge, the emergence of vortex breakdown, and the establishment of a partially premixed front with a stoichiometric ring.

1This work was supported by the National Science Foundation through award 1916979 (Swirling dynamics in liquid-pool fires)

8:26AM C06.00003 Understanding Combustion in the Blue Whirl using Optical Diagnostics. SRIRAM BHARATH HARIHARAN, University of Maryland; Texas A&M University, YEJUN WANG, Texas A&M University, PAUL M. ANDERSON, US Army Research Laboratory, WARUNA D. KULATILAKA, Texas A&M University, MICHAEL J. GOLLNER, University of Maryland, ELAINE S. ORAN, Texas A&M University — The blue whirl is a soot-free flame that evolves from a traditional fire whirl in a regime close to the extinction limit for fire whirls. Previous work considered the physical and thermal structure of the flame and scaling approaches to predict transition to the blue whirl. To better understand the exact nature of combustion in the blue whirl, the present work uses chemiluminescence and planar laser-induced fluorescence (PLIF) techniques to visualize the reaction zone and distribution of radicals in the flame. Chemiluminescence images of excited-state OH* and CH* were obtained at 1 kHz and converted into a radial map using Abel inversion and the OH*/CH* intensity ratio was computed for the region around the bright blue vortex rim. A relationship between this ratio and the local equivalence ratio ($\Phi \in [0.7,1.3]$) was obtained by extrapolating from adiabatic methane-air flames stabilized over a Hencken calibration burner. In addition, PLIF images show that ground-state OH exists in high concentrations in the region of the bright blue ring and diminishes gradually into the post-flame region. Negligible presence of OH was observed in the blue conical region below the ring.

8:39AM C06.00004 On the Consistent Flamelet Model Formulation for Transcritical Fluid Flow and Combustion. ZHENG QIAO, YU LV, Department of Aerospace Engineering, Mississippi State University — Fluid flows in transcritical thermodynamics conditions have numerous applications in combustion engines and thermal energy devices. Under such conditions, the fluid behaviors are largely affected by the peculiar viscous transport properties and nonlinear thermodynamic relations. To achieve cost-effective modeling of transcritical fluids, the flamelet model has been exploited and investigated by a number of studies. This study focuses on the effects of model formulations on the flamelet predictions of transcritical fluids in both pure-mixing and combustion conditions. We first derive the consistent flamelet formulation based on Pitsch & Peters (1998) and then examine the constant Lewis assumption and the interspecies enthalpy flux on the flamelet predictions. For illustration purpose, the H2/air system at various thermal conditions is considered. Our study concluded that it is of importance to consider the detailed consistent model formulation in flamelet modeling of transcritical fluids.

8:52AM C06.00005 Modeling and Simulation of a H2/O2 Diffusion Flame under Cryogenic Condition using a Consistent Flamelet Formulation. YU LV, ZHENG QIAO, Department of Aerospace Engineering, Mississippi State University — Modeling of combustion phenomena under supercritical/transcritical conditions has recently gained growing interest due to the increasing applications of high-pressure combustion and energy devices. Under such thermodynamic conditions, combustion dynamics and flame characteristics are strongly affected by the peculiar thermo-diffusive transport properties and the nonlinear equilibrium-of-state relation. In this study, we first illustrate the effects through a flamelet study in which the comprehensive and consistent flamelet model is employed against the commonly used flamelet model formulation. Then the consistent flamelet model is employed to the LES of a GH2/LOx cryogenic flame based upon a newly developed computational framework. The simulation results will be extensively discussed and compared qualitatively with the experimental measurements by Candel et al. (2006).
8:13AM C07.00002 The 2019 MRV Challenge Experiment at Stanford University¹, CHRISTOPHER J. ELKINS, ANDREW J. BANKO, JOHN K. EATON, Stanford University, MICHAEL J. BENSON, United States Military Academy — Specific measurement techniques and Magnetic Resonance Velocimetry (MRV) results for the 2019 MRV Challenge experiment at Stanford University will be presented in detail. Some basic physics behind MRV as well as a general description of how pulse sequences (series of radio frequency pulses and applied magnetic field gradients) are used to make velocity measurements will be introduced. This will provide context for the motivation behind the challenge and the differences observed among the measurements by the participating labs since each lab uses a different pulse sequence in their own MRI system. Specific to the Stanford experiment, the equipment used and the overall setup of the square cross-section U-bend channel in the 3T magnet at the Richard M. Lucas Center for MRI at Stanford will be described. In addition, the parameters of the MRV pulse sequence and post processing steps of the data will be explained. Finally, an overview of the results and uncertainties will be given along with a discussion of the sources of these uncertainties.

¹Department of Defense

8:26AM C07.00003 2019 MRV Challenge: Hanyang University and Korea Basic Science Institute Results. , SIMON SONG, MUHAMMAD HAFIDZ ARIFFUDIN, DON-GWAN AN, CHAEHYUK IM, Hanyang University, SUKHOON OH, Korea Basic Science Institute — This presentation is part of the 2019 MRV Challenge, and represents the results of a combined team from Hanyang Univ. and KBSI. Four MRI research groups are supposed to make measurements in the same apparatus comprising a square cross section U-bend with a tight radius that will produce a highly three-dimensional flow field with turbulent flow separation. An inlet boundary layer trip marks the common coordinate origin of the flow in the channel. The apparatus being transferred between the research labs includes detailed flow conditioning to ensure that variations in supply and exit plumbing will not disrupt the test section flow. The test was conducted at a turbulent Reynolds number of 15,000 based on the hydraulic diameter. The presentation will focus on the methodology used including the MRI hardware and software, parameters for measurement, as well as the number of scans and overall scan duration, software sequence details, as well as post-processing and filtering techniques. In addition, details of the flow at several locations will be presented, along with estimates of uncertainty for each velocity component. Finally, an estimate of experimental effort — comprised of number of personnel involved and hours, costs, and other factors will be provided.

8:39AM C07.00004 Magnetic Resonance Velocimetry in High-Speed Turbulent Flows - Sources of Measurement Errors and a New Approach for Higher Accuracy , MARTIN BRUSCHEWSKI, KRISTINE JOHN, SVEN GRUNDMANN, University of Rostock — Magnetic Resonance Velocimetry (MRV) has great potential to become a versatile velocity measurement technique for applied fluid mechanics. One of the most dominant errors in MRV is the effect of displacement which describes the spatial misregistration of the acquired signal in a moving fluid. The overall aim of this study is to highlight the significance of displacement errors in conventional MRV and to provide an improved method. A new MRV sequence, named SYNC SPI (single point imaging with synchronized encoding) has been developed to significantly reduce this error. Measurements were performed in several test cases including a U-bend as part of the 2019 MRV Challenge. In comparison to conventional MRV, this sequence provides reliable velocity data for a wide range of flow velocities. It is shown that flow velocities up to 15 m/s can be accurately measured with this technique. The main disadvantage of the SYNC SPI sequence is the relatively long acquisition time. This disadvantage is partly resolved using a modern undersampling technique called Compressed Sensing to reduce the number of samples required to provide fully-resolved velocity data. It is shown that the acquisition time can be reduced by more than 70% while still maintaining high measurement accuracy.

8:52AM C07.00005 2019 MRV Challenge: Mayo Clinic Results¹, DANIEL BORUP, Mayo Clinic — This presentation is part of the 2019 MRV Challenge focus session. The results presented were obtained on a Philips 3T Elition X scanner. The flow geometry consisted of a U-bend with square cross section. The flow was fully turbulent with a bulk Reynolds number of 15,000. The flow loop was assembled and operated as described in the MRV Challenge instructions. This presentation will focus on details of the measurement including the choice of scan parameters, sequence design, and the overall measurement setup used to obtain data. The results from standard 3D Cartesian-trajectory imaging will be presented as a baseline, while data obtained in a shorter time using a “spiral readout” trajectory will be shown for comparison. Time-series data in 2D planes in the bend will also be presented to highlight any observations regarding the flow unsteadiness. The measurement uncertainties will also be presented.

¹Research Support from Philips Healthcare

9:05AM C07.00006 2019 MRV Challenge: Results and Comparisons, MICHAEL BENSON, U.S. Military Academy, CHRISTOPHER ELKINS, ANDREW BANKO, Stanford University, SIMON SONG, Hanyang University, SVEN GRUNDMANN, MARTIN BRUSCHEWSKI, University of Rostock, DANIEL BORUP, Mayo Clinic — This presentation is part of the 2019 Magnetic Resonance Velocimetry (MRV) Challenge and represents the combined results from all the participants of the challenge. Four research groups have made measurements in the same apparatus comprising a square cross section U-bend with a tight radius that will cause turbulent flow separation. An inlet boundary layer trip marks the common coordinate origin of the flow in the channel. The apparatus that was transferred between the research labs includes detailed flow conditioning to ensure that variations in supply and exit plumbing will not disrupt the test section flow. The test was conducted at a Reynolds Number of 15,000 based on the channel hydraulic diameter, and a tight radius U-Bend ensures a strongly three-dimensional flow field. This presentation will focus on the results from each team, which will be compared in the region near the boundary layer trip, at the entrance to and through the U-Bend, and through the exit passageway using velocity field contour plots, iso-surfaces, and quantities derived from the mean velocity components. Similarities and differences from the results will be presented which can provide insight into the opportunities that state of the art MRV provides to researchers.

9:18AM C07.00007 Investigation of Gas Turbine Cooling Flows Using Magnetic Resonance Velocimetry¹, WONTAE HWANG, SEUNGCHAN BAEK, SANGJOON LEE, JAEHYUN RYU, Seoul National University — Gas turbine blades are exposed to extreme high temperature, beyond the melting point of the material, thus various methods of advanced cooling are applied. Internal cooling extracts heat from the blade surface. Internal cooling within multiple channels inside the blade. These complex channels are spaced tightly together, and it is difficult to perform optical measurements of the flow. To overcome this problem, Magnetic Resonance Velocimetry (MRV) can be utilized to obtain the 3D flow field within these cooling channels. In this study, we demonstrate how we use MRV to analyze the complex flow field within turbine blade cooling structures such as a trailing edge internal channel which has rib turbulators and a thin sharp corner. The data can be used to provide quantitative validation for RANS and LES CFD.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2017R1A2B4007372 and 2017R1A1A1015523)
Organizes the buffer layer structures near both upper and lower walls. Structures near the wall are more correlated with those near the opposite wall in DR flow, which implies that a large scale outer structure in the streamwise direction. Premultiplied two-dimensional spectra of streamwise velocity fluctuations show that the outer turbulence structure by adding polymer is modeled by the FENE-P model. It is found that in the DR flow, the length scale significantly increases, especially in the spacing between the volute wall and the blades is very small. Phase-locked experiments were conducted at the lower Reynolds number to ensure best measurement quality, yielding phase-locked velocity fields between the blades at five positions within a blade passing cycle. Alternating jet-wake structures are observed protruding from each blade passage around the impeller simultaneously. The varying size and shape of separation bubbles on the suction side of the blades is also analyzed as they rotate through an entire revolution.

Support by DENSO Corporation

9:44AM C07.00009 Mean Flow Field Measurements in Cavitating Flow Using Magnetic Resonance Velocimetry Supported by X-Ray and Particle Image Velocimetry

KRISTINE JOHN, MARTIN BRUSCHEWSKI, University of Rostock, SAAD JAHANGIR, WILLIAM HOGENDOORN, EVERT C. WAGNER, ROBERT F. MUDDE, CHRISTIAN POELMA, Technische Universiteit Delft, SVEN GRUNDMANN, University of Rostock — This presentation focusses on mean flow field measurements in cavitating flow using Magnetic Resonance Velocimetry (MRV). An intrinsic feature of MRV is its ability to measure without optical access. Optical refractions at the phase changes between liquid and vapor phases, which render optical data unusable, do not affect MRV. Moreover, the density of the nuclear spins that is used as a signal source for MRV is typically much higher inside the liquid phase. Voids based on signal intensity and measurements of the mean flow velocities in the liquid phase are possible. For a proof-of-concept, flow cavitation in a venturi is investigated. The MRV data is validated with PIV data up to the point of cavitation. For the cavitating cases, X-Ray measurements of the void mean fraction are used as support. It is shown that MRV can provide reliable velocity data. Quantitative void fraction measurements based on the signal intensity of MRV are cumbersome. Flow effects such as turbulence attenuate the signal intensity, which cannot be distinguished from signal voids caused by the vapor phase. The velocity data from MRV must be supported by void fraction data such as from X-Ray. Together, these two techniques provide a valuable tool for studies in cavitating flow.

9:57AM C07.00010 Purely Phase-Encoded Magnetic Resonance Mapping of Turbulence Anisotropy

BENEDICT NEWLING, AMY-RAE GAUTHIER, ALEXANDER ADAIR, Department of Physics, University of New Brunswick — The measurement of flow velocities much greater than 1 m/s can be a challenge for conventional magnetic resonance imaging (MRI) methods. Motion during the spatial encoding interval can lead to a variety of geometric and anemometric distortions. Purely phase-encoded MRI methods, such as SPRITE (single-point ramped imaging with T1 enhancement) can employ a short encoding interval (hundreds of microseconds) for the time-averaged measurement of fast flows. The interval is not only short, but also constant, which saves SPRITE from artefacts caused by interfaces in multi-phase flow. Most recently, we have been using SPRITE to measure mean-squared displacements in turbulent flow in order to quantify the anisotropy of velocity fluctuations. By analogy with diffusion tensor imaging, we measure the components of an eddy self-diffusivity tensor downstream of a Venturi constriction at Reynolds numbers on the order of 105.

Work supported by the Natural Sciences and Engineering Research Council of Canada

Sunday, November 24, 2019 8:00AM - 10:10AM
Session C08 Non Newtonian Flows : Instability and Turbulence 212 - Satish Kumar, University of Minnesota

8:00AM C08.00001 Modification of vortex shedding and turbulence statistics in a two-dimensional turbulent flow affected by polymers.

RURI HIDEMA, KENGO FUKUSHIMA, HIROSHI SUZUKI, Department of Chemical Science and Engineering, Kobe University. — FLUIDS AND PARTICLE ENGINEERING LABORATORY TEAM — An experimental study was performed to investigate the relationship between the extensional rheological properties of polymer solutions and vortex deformation in turbulent flow. In order to focus on the effect of extensional rheological properties of the fluids, polyethyleneoxide was added to two-dimensional (2D) turbulent flow. The 2D flow was self-standing flowing soap films that is relatively free from shear stresses. Therefore, the flow is advantageous as it examines the effect of the extensional rheological properties of polymers on the flow. In the study, the vortex shedding in 2D turbulent flow and turbulence statistics of the vortices in the flow were observed using interference patterns and particle image velocimetry (PIV). We found that the vortex shedding in the 2D flow was categorized into three types, and this was affected by the relaxation time of the polymer solutions. The modification of the vortices varied the energy transfer in 2D flows. We have also found that characteristic energy peak due to extensional rheology of the fluids.

A Grant-in-Aid for Scientific Research (B) (Project No.: 19H02497), Japan Society for the Promotion of Science.

8:13AM C08.00002 Organization of turbulence structures in polymer drag-reduced turbulent channel flow

KYOUNGYOUN KIM, Hanbat National University — To investigate turbulence structures in polymer drag-reduced (DR) channel flows, the spectra and correlations of the velocity fluctuations are examined by performing direct numerical simulations of viscoelastic turbulent channel flows of a drag reduction rate of 63%. The friction Reynolds number is Re = 395 and the stresses created by adding polymer is modeled by the FENE-P model. It is found that in the DR flow, the length scale significantly increases, especially in the streamwise direction. Premultiplied two-dimensional spectra of streamwise velocity fluctuations show that the outer turbulence structure in the DR flow differs from those in the Newtonian flow. Two-point correlations and conditional average flow fields reveal that the buffer layer structures near the wall are more correlated with those near the opposite wall in DR flow, which implies that a large scale outer structure organizes the buffer layer structures near both upper and lower walls.
8:26AM C08.00003 Turbulent Boundary-Layer of Power-law Fluids Near a Position of Separation. JULIANA LOUREIRO, ATILA SILVA FREIRE, Federal University of Rio de Janeiro — The study describes the turbulent boundary layer structure near and at a separation point for power-law fluid flows. Experimental work is performed for flow of water and a 0.1% carboxymethyl cellulose (CMC) water blend, over an asymmetric plane diffuser with 30-degree slope. The flow index $n$ and the consistency parameter $K$ are respectively 0.86 and 0.00753 (Pa s$^n$). Particle Image Velocimetry and Laser Doppler Velocimetry are used to introduce profiles of local mean velocity, turbulent shear stresses at the points of separation, reattachment and in the recirculation region. The location of the separation and reattachment points are described in terms of changes in the generalized Reynolds number based on the channel height and the shear stress conducted for finite Reynolds numbers. The work reports large changes in the length of the separation regions and discusses the local solutions of Goldstein (1948) and Stratford (1959). In the fully viscous region, the mean velocity is shown to vary as $y^{(n+1)/n}$, as expected from the local analytical solution, with $y$ as the distance from the wall. In the fully turbulent region, the mean velocity profile follows a $y^{1/2}$ law, being thus independent of rheology of the fluid.

8:39AM C08.00004 ABSTRACT WITHDRAWN —

8:52AM C08.00005 Exact coherent structures in two-dimensional viscoelastic channel flow. JACOB PAGE, University of Cambridge, YVES DUBIEF, University of Vermont, RICH KERSWELL, University of Cambridge — Elasto-inertial turbulence (EIT) is a recently discovered flow state in dilute polymer solutions that is strikingly different to quasi-Newtonian, drag-reduced flows. EIT is largely two-dimensional; its dominant flow features are thin sheet-like structures of polymer stress with attached patches of intense spanwise vorticity. In this talk we explore the mechanics underpinning EIT by searching for exact coherent structures in elasto-inertial planar channel flows. The structures we find are all connected to a recently discovered linear instability (Garg et al, Phys. Rev. Lett. 121, 2018) of the basic state that exists at moderate elasticities $Wi < Re < \mu^3$. Just beyond the point of marginal stability ($Wi \approx 25$, $Re \sim 50$) the universal attractor is a relative periodic orbit (RPO) featuring a pair of large amplitude sheets that meet at the centreline. This RPO bifurcates off a strongly subcritical travelling wave that exists for Weissenberg numbers as low as $Wi \sim 10$. We perform branch continuation of this travelling wave upwards in Reynolds number to explore its overlap with EIT. Time permitting, we will also discuss the physical mechanisms at play in the new linear instability.

9:05AM C08.00006 Critical layers, Tollmien-Schlichting waves and elasto-inertial turbulence. ASHWIN SHEKAR, University of Wisconsin-Madison, RYAN MCMULLEN, BEVERLEY MCKEON, Caltech, MICHAEL GRAHAM, University of Wisconsin-Madison — We describe direct numerical simulations (DNS) and linearized analyses of channel flow turbulence in a FENE-P fluid in the elasto-inertial turbulence (EIT) regime. Simulations at low (transitional) Reynolds numbers are shown to display localized polynomial stretch fluctuations very similar to structures arising from linear stability (Tollmien-Schlichting (TS) modes) and resolvent analyses: i.e., critical-layer structures localized where the mean fluid velocity equals the wavespeed. Self-sustained nonlinear TS waves display stagnation points that generate sheets of large polymer stretch. These kinematics may be the genesis of similar structures in EIT. We also describe a new self-sustaining elastoinertial state which we term the lower branch attractor (LBA), which has very small amplitude and whose structure closely resembles that of the linear TS mode. A tentative bifurcation scenario describing our observations is described. We also identify the minimal flow unit for EIT, which at low Reynolds number continues to exhibit localized stress fluctuations. Finally, resolvent analysis and transient simulations of the linearized problem shed light on the origins of the mechanisms leading to amplification of fluctuations and thus to the bypass-transition nature of the onset of EIT.

8:18AM C08.00007 What do we mean by the mean conformation tensor (in viscoelastic turbulence)? ISMAIL HAMEDUDDIN, Norton-Rose-Fullbright, TAMER ZAKI, Johns Hopkins University — The popular arithmetic mean conformation tensor frequently used in the analysis of turbulent viscoelastic flows is not a good representative of the ensemble. Alternative means more faithful to the tensorial character of the conformation tensor are evaluated, namely, the geometric and square root of the Reynolds number. It is noteworthy that this primary instability is non-hysteretic and appears in the absence of perturbations of the elastic, viscous, and inertial forces on the ejected droplet size. These simulations serve as a departure point for further work involving three-dimensional simulations of atomisation processes of viscoelastic jets.


9:31AM C08.00008 En Route to the Maximum Drag Reduction Asymptote. GEORGE CHOUEIR, JOSE LOPEZ, ATUL VARSHNEY, BJORN HOF, IST Austria — We report the results of an experimental investigation into the stability and transition of a supersonic pipe flow. Wave stability is investigated at different Reynolds number $O(1)$, provided that the shear stress (and hence the Weissenberg number) is sufficiently large, all the way to the maximum drag reduction (MDR) asymptote for dilute polymer solutions. At onset the amplitude of streamwise velocity fluctuations is found to grow with the square of the Reynolds number. It is noteworthy that this primary instability is non-hysteretic and appears in the absence of perturbations and curved streamlines. The flow structures observed closely resemble those of the unstable mode discovered in a recent stability analysis. Further from onset, a secondary instability is found where the azimuthal symmetry is broken and inclined streaks of high amplitude appear in the near wall region. These streak patterns resemble the typical flow structures found in MDR turbulence at higher Reynolds numbers.

9:44AM C08.00009 Two-dimensional direct numerical simulations of viscoelastic jets. KONSTANTINOS ZINELIS, THOMAS ABDIE, RICARDO CONSTATE-AMORES, OMAR MATAR, Imperial College London — The numerical simulation of spray formation in a non-Newtonian fluid offers substantial challenges and is central to numerous industrial applications such as spray-drying. The aim of the present work is to set the basis for the numerical examination of non-Newtonian atomisation and spray systems. To achieve this, a Direct Numerical Simulations (DNS) approach is followed where all the temporal and spatial scales are resolved completely. We begin with the simulation of two-dimensional numerical simulations of an Oldroyd-B impulse jet as well as a jet with a constant release velocity into a stagnant gaseous phase using the volume-of-fluid technique to capture the interface and the S-configuration transformation for the solution of the viscoelastic constitutive equation. This permits the exploration of parameter space, capturing the effect of the elastic, viscous, and inertial forces on the ejected droplet size. These simulations serve as a departure point for further work involving three-dimensional simulations of atomisation processes of viscoelastic jets.

1Engineering and Physical Sciences Research Council, UK, i-case studentship (sponsored by Johnson Matthey) for Konstantinos Zinelis
9:57AM C08.00010 A Well-Conditioned Numerical Method for Resolvent Analysis of Viscoelastic Channel Flows1, GOKUL HARIHARAN, Department of Chemical Engineering and Materials Science, University of Minnesota, MIHAILO JOVANOVIC, Ming Hsieh Department of Electrical and Computer Engineering, University of Southern California, SATISH KUMAR, Department of Chemical Engineering and Materials Science, University of Minnesota — Linear analyses provide useful information about the potential for transition to nonlinear states. While a modal approach furnishes information about long-time growth or decay of initial conditions, non-modal approaches give insight into the amplification of disturbances in a linearly stable flow. Here, we conduct non-modal analysis of inertialless 2D viscoelastic channel flows. Our analysis reveals large stress gradients in the near-wall region (for plane Poiseuille flow) and in the channel center (for plane Couette flow). These steep stress gradients can only be resolved using recently developed well-conditioned spectral methods, e.g., the ultraspherical and spectral integration methods. Furthermore, even if the discretization method is well-conditioned, computation of frequency-responses can be erroneous if singular values are obtained as the eigenvalues of a cascade connection of the resolvent operator with its adjoint. We address this issue by introducing a feedback interconnected system that avoids matrix inverses and allows reliable frequency-response calculations of viscoelastic channel flows at high Weissenberg numbers (~500). The steep stress gradients that we identify may play a role in explaining recent experiments concerning transition to elastic turbulence.

1National Science Foundation

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C09 Turbines: General 213 - Brent Houchens, Sandia National Laboratories

8:00AM C09.00001 Influence of Near-blade Hydrodynamics on Cross-flow Turbine Performance, ABIGALE SNOTLAND, BRIAN POLAGYE, OWEN WILLIAMS, University of Washington — Cross-Flow turbines are a promising technology for harvesting kinetic energy from wind and water currents. The hydrodynamics are complex and rapid changes in angle of attack lead to phenomena such as dynamic stall and periodic vortex shedding. A robust understanding of local flow features at the turbine blades and their impact on phase-averaged performance is essential for turbine design and can inform control strategy development. Phase-averaged 2D planar PIV data is examined for both optimal (maximum power generation) and sub-optimal rotation rates over the entire sweep of a two-bladed, cross-flow turbine. The phase and rotation-rate dependent, near-blade/wake hydrodynamics are examined in concert with phase-averaged performance data. The duration/severity of flow reversal on, and detachment from, the blades appear critical to average performance. Strong stall and vortex interactions occur over most of the turbine rotation for the sub-optimal case, possibly producing parasitic forces that outweigh any increases in lift from the leading-edge vortex. At the optimal rotation rate, vortex shedding on the upstream blade is significantly delayed, and the downstream blade is weakly stalled. This likely explains the difference in power output between the two cases.

8:13AM C09.00002 Improving the accuracy of wind farm LES using filtered actuator disk theory corrections1, CARL SHAPIRO, DENNICE GAYME, CHARLES MENEVEAU, Johns Hopkins University — The utility of large eddy simulations (LES) of large wind farms that employ the actuator disk model (ADM) are limited by over-prediction of power that can exceed 10% at typical resolutions. Computational restrictions require spatial filtering of the actuator disk thrust force, which is distributed equally across the swept area of the rotor blades, resulting in an under-prediction of the shed vorticity. The filtered ADM, which models the wind turbine wake as concentric semi-infinite vortex cylinders, provides a basis for analytically correcting this error in simulation. When compared to simulations with various filter widths and grid sizes, the filtered ADM accurately predicts the power coefficient measured in simulations. An analytic correction factor is then derived from the filtered ADM that collapses the power coefficient measured in simulations onto the theoretical axial momentum theory predictions. This approach eliminates the need for highly refined numerical grids or empirical correction factors.

1Funded by NSF (CMMI 1635430). Computations performed using MARCC resources.

8:26AM C09.00003 LES of microscale winds in complex configurations1, LAURENT BRICTEUX, STEPHANIE ZEOLI, MONCEF GAZDALLAH, University of Mons, PAULINA PANKO-OPRYCH, West Pomeranian University of Technology, Szczecin, GREGOIRE LEROY, 3E, PIERRE BENARD, INSa Rouen Normandie / CORIA — This contribution concerns wall modeled large eddy simulations (WMLES) of turbulent winds in complex geometries that are relevant for wind energy problems: either in the built environment (e.g. generic buildings) or complex terrains (e.g. Askervein Hill). This work is performed in the framework of developing simplified operational models to be used for wind turbine siting. It is proposed to investigate the influence of the wall modeling strategy and the turbulent inflow boundary condition on the flow diagnostics relevant for wind turbine siting (local velocity and turbulence profiles). The validity and quality of an original turbulence inflow generation method are also assessed. This boundary condition is based on the algorithm of Mann and generates realistic atmospheric turbulence with a prescribed spectrum and lengthscale.

1Belgian Walloon Region, Mecatech cluster, Pope project

8:39AM C09.00004 Power measurements in a scale model wind farm: Results from varying array size, arrangement, spacing, and setpoints, DAN HOUCK, EDWIN COWEN, Cornell University — The current trend in wind energy research is to optimize wind farms as opposed to optimizing individual turbines. There is also an emerging idea to consider the wind turbines themselves as actuators that can be used to intentionally and beneficially manipulate the flow to improve the power output of the wind farm. To this end, we completed a series of experiments with an array of 18 model-scale wind turbines in a 2 m wide flume testing changes in the number, arrangement, and spacing of the turbines as well as the setpoint, or power production, of each turbine. Each treatment case is compared to a similar control case that was arranged and operated more conventionally with all turbines attempting maximum power production. A highly accurate torque transducer provides calibrations allowing non-intrusive mechanical power measurements of each turbine. Comparisons are made on the basis of overall power output, array efficiency (total power output of the N-turbine array divided by N of an upstream turbine operating at max power), and power density (power per area). Particle image velocimetry (PIV) further reveals the fluid dynamics at work to create any improvements in power.
A novel wind energy harvester with no external moving parts is demonstrated at one-third scale (0.5-meter chord) in wind tunnel tests. The device uses mirrored airfoil-pairs to create suction. This pulls air out from ducts internal to the foils, through air-jet orifices on the low-pressure sides of the foils, and into the external incident wind. Power is transmitted pneumatically through the center of the foils to an internal turbine-generator. In the high-performance operating mode at high angle-of-attack, a mechanical power transmission of nearly one-half of the Betz limit is achieved. However, this high angle-of-attack configuration is susceptible to aero-acoustic instabilities which can diminish the performance to as low as one-sixth of the Betz limit. These instabilities are investigated here for a configuration with all air-jets covered. Pressure measurements are made on the low-pressure sides of the foils and the break in symmetry associated with the instability is documented. Particle image velocimetry is used to characterize the flow field before and after onset of the instability. SNL is managed and operated by NTI under DOE NNSA contract DE-NA0003525.

9:05AM C09.00006 Experimental comparison of HAWTs with hydrostatic and regular transmissions, HELBER ANTONIO ESQUIVEL-PUENTES, ANDREA VACCA, School of Mechanical Engineering, Purdue University, LEONARDO P. CHAMORRO, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, JOSE GARCIA-BRAVO, School of Engineering Technology, Purdue University, HUMBERTO BOCANEGRA-EVANS, DAVID WARSINGER, WALTER GUTIERREZ, LUCIANO CASTILLO, School of Mechanical Engineering, Purdue University — The performance of a horizontal axis wind turbine (HAWT) with hydrostatic transmission was compared with that of a standard unit using well-controlled experiments. The power output of the hydrostatic unit is used for electricity generation, and/or reverse osmosis to obtain fresh water. Furthermore, the hydrostatic transmission allows for moving the major electro-mechanical components located in the hub to the ground level. The associated changes resulted in reductions of approximately 30% of the total mass of the unit, and of about 7 to 14% of the cost of the energy. The reconfiguration of the structure of the turbine also implied changes in the response of the system to turbulence. We assessed such effects across scales considering changes in the transfer function of the spectrum of the power output, following Tobin et al. (2015).

9:18AM C09.00007 Reliability of Long-term Lidar-based Wind Measurements for Various Wind Energy Applications, JAY PRAKASH GOIT, Department of Mechanical Engineering, Kindai University, Japan, SUSUMU SHIMADA, TETSUYA KOGAKI, National Institute of Advanced Industrial Science and Technology, Japan — In this work, we conduct a long-term measurement campaign (approximately one year) using a profiling Lidar, in order to investigate whether Lidars can be used as a reliable alternative to meteorological masts for wind energy applications. It is found that the Lidar-measured wind speed showed good agreement with those measured using sonic anemometers mounted to the neighboring meteorological mast, with the coefficient of determination ($R^2 > 0.99$). However, comparison of the standard deviations shows larger degree of variations, with $R^2$ between 0.92 and 0.97. Turbulence intensities computed for the 90th percentile of the standard deviation show that the Lidar-measured turbulence intensities are larger by roughly 2% than those measured by the sonic anemeter. But the gust factors for peak wind speeds converge roughly to 1.9 during strong wind speed for both the devices. Finally, wind speed and turbulence distribution for the Lidar and sonic anemometer are used to compute fatigue load for NREL 5-MW reference wind turbine using aerelastic simulation. The 20 years lifetime DELs for the Lidar wind speed are higher than those for the sonic anemometer wind speeds, by 6% for the blade root bending moment and by 13% for the tower based bending moment.

9:31AM C09.00008 Actuator-line simulations of wind turbines with block-structured adaptive mesh refinement, MAHESH NATARAJAN, HARISWARAN SITARAMAN, SHREYAS ANANTHAN, MICHAEL ALAN SPRAGUE, National Renewable Energy Laboratory — Wind turbine parametrization methods such as the actuator disk (ADM) or actuator line methods (ALM) have shown good promise in determining power generation and loads on the turbine blades. However, performing such simulations in a cost-effective manner is a challenging task, due to the wide range of scales involved. The scales in the atmospheric boundary layer (ABL) range from 1 km to scales in the wake region that are 1 m. Adaptive mesh refinement techniques are well suited for this scenario, and will enable well-resolved simulations that can capture turbulent structures across multiple length scales. In this work, we implement an ALM model in two frameworks - a compressible and an incompressible flow solver - both developed within the block-structured adaptive mesh refinement (AMR) framework of AMReX [1], and compare their performance. Computational scaling studies are done on a single turbine configuration, and compared with Nalu-Wind [2] - an unstructured CFD solver for wind turbine simulations.

1This work was funded by the U.S. Department of Energy under Contract No. DE-AC36-08-GO28308 with the National Renewable Energy Laboratory.
2Researcher II - Computational Science
3Researcher IV - Mechanical Engineering
4Researcher V - Software Engineering
5Researcher VI - Computational Science

9:44AM C09.00009 Biomimetic individual pitch control for wind turbines, MARION COQUELET, UCLouvain - UMONS, LAURENT BRICTEUX, Universite de Mons (UMONS), MAXIME LEJEUNE, PHILIPPE CHATELAIN, Universite catholique de Louvain (UCLouvain) — Individual Pitch Control (IPC) has proved efficient in reducing fatigue loads on wind turbines. As 1P (once-per-revolution) blade loads are significant due to wind shear or tower shadow effect, their alleviation implies a rhythmic behavior in the changes of the blade pitch angles. This work relies on Large Eddy Simulations (LES) to demonstrate the effectiveness of a biomimetic architecture for IPC. The high-fidelity simulation of the flow physics is essential to assess the performances of the controller. It is performed by an in-house lifting-lines-enabled Vortex Particle-Mesh method, dealing with synthetic turbulence and wind shear at the inflow. The individual pitch controller is based on Central Pattern Generators (CPGs). CPGs produce, from very simple inputs, rhythmic outputs that drive repetitive motions like walking or breathing. The specificity of these networks is their ability to operate autonomously. They are here implemented as coupled non-linear oscillators, capable of producing coordinated rhythmic patterns and driven by a knowledge of the upstream flow conditions. The latter are reconstructed online by an Extended Kalman Filter processing the loads experienced by the individual blades.

1Funding from the European Research Council under the European Unions Horizon 2020 research program. Computational resources made available on the Tier-1 supercomputer funded by the Walloon Region.
The mean velocity is plotted in defect formulation, i.e., $U - U(\eta_m)/\eta_m$. The intermediate-scaled mean velocity and variance exhibit Reynolds-number invariance over a certain region around $\eta = 1$. This implies existence of a Reynolds-number independent log law in terms of the intermediate variables for the mean velocity and variance. The classical “Karman constant” and “Townsend-Perry constant” are expressed in terms of the log-law constants obtained from the intermediate scaling and their dependence on Reynolds number examined. The present work suggests a three-layer asymptotic structure for the TBL, with each layer governed by distinct length and velocity scales, which results in two overlap layers (Afzal. Ing.-Arch. 1982). We discuss the relevance of the power/log law behavior of the mean velocity in the overlap layers, based on the ratio of the velocity scales of the adjacent layers.

We acknowledge financial support from EPSRC (EP/1037938/1).

**8:13AM C10.00002 Log-law & power-law: local self-similarity of a flat-plate boundary layer**

JOSEPH RUAN, GUILLAUME BLANQUART, California Institute of Technology — A large controversy in the previous decade for flat-plate boundary layers was on the scaling of the defect layer with respect to different velocity scales, made prominent by George & Castillo (1997). The lack of enough large Reynolds number experiments and simulations makes the choice of appropriate velocity scale rather difficult. We show mathematically that the differences between the scaling descriptions (using the free-stream velocity or the friction velocity as a scaling parameter) may in fact be negligible for local regions of the boundary layer. We verify from a variety of simulation frameworks that the results from filtered-lifting-line-theory and LES using the actuator line model are in excellent agreement, as expected, and agree well with the measurements. The results suggest that when computing the forces along the blade, filtered-lifting-line-theory, LES using the actuator line model, and blade resolved DES agree well with the experiment and the differences between each method are small. However, when comparing the flow fields downstream, blade resolved simulations are able to accurately capture the tip-vortex, whereas the actuator line model cannot always capture the flow details from the tip vortex.

**8:26AM C10.00003 A Spectral-Scaling Based Extension to the Attached Eddy Model of Wall-Turbulence**

DILEEP CHANDRAN, JASON MONTY, IVAN MARUSIC, The University of Melbourne — An extension to the attached eddy model (AEM) for the logarithmic region of turbulent boundary layers is presented here. The extension is driven by the scaling of two-dimensional (2-D) spectra of the streamwise velocity component, measured at friction Reynolds numbers ranging from 2400 to 26000. The conventional AEM assumes the boundary layer to be populated with hierarchies of self-similar wall-attached (TypeA) eddies alone. While TypeA eddies represent the dominant energetic large-scale motions at high Reynolds numbers, the scales that are not represented by such eddies are observed to carry a significant proportion of the total kinetic energy. Therefore, in the present study, we propose an extended AEM that incorporates two additional representative eddies. These eddies, named TypeCA and TypeSS, represent the self-similar but wall-detached low-Reynolds number features, and the non-self-similar wall-attached structures, respectively. The extended AEM is observed to predict reasonably well a greater range of energetic length scales and capture the low- and high-Reynolds number scaling trends in the 2-D spectra of all three velocity components.

**8:39AM C10.00004 Spatio-Temporal Characteristics of Coherent Structures in Shear-Dominated Flows**

TAYGUN RECEP GUNGOR, AYSE GUL GUNGOR, Istanbul Technical University, YVAN MA-CIEL, Universit Laval, MARK PHIL SIMENS, Universidad Politecnica de Madrid — The energy and Reynolds stress carrying structures are investigated using three different numerical simulation databases. The first and second databases are non-equilibrium adverse gradient pressure (APG) turbulent boundary layers (TBLs) with $Re_{\theta}$ reaching 8000. In the second case, the turbulent activity in the inner layer ($y/\delta < 0.1$) is artificially eliminated to examine outer of layer APG TBLs in the absence of near-wall turbulent activity. The last one is a homogeneous shear flow (HSF) database that provides information about a shear dominated flow without a wall. The turbulence statistics and structures in three flow cases are compared to understand similarities and differences between the outer layer of APG TBLs and HSF. Results of the manipulated APG TBL indicate that outer layer turbulence sustains itself when there is no turbulence activity in the inner layer and the spatial-temporal characteristics of the energetic structures are similar to the structures found in the original APG TBL. Furthermore, Reynolds stress carrying structures in the APG TBLs resemble the ones in HSF when their dimensions are scaled with the Corrsin length scale.

Funded in part by ITU BAP, NSERC of Canada and the Coturb program of the European Research Council.
Reynolds number dependence of turbulence statistics near the turbulent/non-turbulent interfacial layer in turbulent boundary layer. XINXIAN ZHANG, TO-MOAKI WATANABE, KOJI NAGATA, Nagoya University — Direct numerical simulations (DNS) of a temporally developing turbulent boundary layer (TBL) are performed for investigating the Reynolds dependence of the turbulent/non-turbulent interfacial layer (TNTI layer). The Reynolds number based on the momentum thickness ranges from 2000 to 13000 in the present DNS. The outer edge of the TNTI layer, called the irrotational boundary, is detected as an isosurface of vorticity magnitude, and the conditional statistics are calculated conditioned on the distance from the irrotational boundary. The results show that the mean thickness of the TNTI layer divided by the Kolmogorov scale is almost constant for different Reynolds numbers when the Kolmogorov scale is taken near the TNTI layer. On the other hand, the mean thickness normalized by Taylor microscale decreases as the Reynolds number increases. Influence of the wall on the statistics near the TNTI layer are shown to be stronger for a lower Reynolds number. Geometry of the irrotational boundary is also studied for the mean curvature and surface area. It is shown that the mean curvature normalized by the Kolmogorov scale has a similar probability density function for all the Reynolds numbers while the surface area increases with the Reynolds number.

Where is the wall? RICARDO GARCIA-MAYORAL, JOSEPH IBRAHIM, GARAZI GOMEZ-DE-SEGURA, University of Cambridge — Textured surfaces with small texture size can increase or reduce turbulent drag, compared to a smooth wall, by imposing different virtual origins for the different streamwise and spanwise components. We extend this idea by imposing different virtual origins for all three velocities using Robin, slip-length-like boundary conditions in direct numerical simulations. We show that the change in drag depends only on the offset between the virtual origin for the mean velocity profile, typically set by the streamwise slip length, and the virtual origin for turbulence (embodied by the quasi-streamwise vortices of the near-wall core), set by the wall-normal and spanwise slip lengths. We demonstrate that, other than by the offset between these origins, turbulence remains essentially smooth-wall-like, and show how to obtain the position of the virtual origins for the mean velocity and for turbulence from the three slip-length coefficients, and from them the change in drag.

Quantifying Reynolds stresses in the planar transitional T3-serious flows. FAN TANG, WEI-TAO BI, ZHEN-SU SHE, State Key Lab for Turb. & Complex Sys., College of Engg., Peking Univ. — The transitional flow with a rapid change of friction coefficient in the streamwise direction is a subject of enormous technological impact, but difficult to quantify, even for simple geometry such as flow passing a smooth flat plate. Recently, a comprehensive theory of turbulent boundary layer is constructed via a Lie-group symmetry approach, yielding a multi-layer description of four stress lengths in the wall-normal direction. Here, we report its extension to transitional TBL, with a unified quantitative description for both the Reynolds shear stress and normal stresses (turbulence intensities) throughout the transition region. Specifically, the transition and subsequent evolution to fully developed TBL are quantified with a streamwise multi-layer description (starting from the leading edge) of two key parameters (a near-wall eddy length, and kappa — a bulk flow parameter), which display a scaling change from laminar to turbulent regime. For the T3-serious planar transitional flows with varying incoming turbulence intensity and pressure gradient covering both natural and bypass transitions, the new theory predicts simultaneously, for the first time, the friction coefficient and wall-normal mean (velocity and turbulent kinetic energy) profiles throughout the entire flow domain. In summary, all four Reynolds stress components are successfully predicted for transitional planar boundary layer, promising future simular and more accurate alternative model for engineering applications.

Near-asymptotics overlap solutions for transport moments in turbulent channels and boundary layers. WILLIAM GEORGE, Dept. of Aeronautics, Imperial College of London, London, UK, JEAN-MARC FOUCAUT, JEAN-PHILIPPE LAVAL, Univ. Lille, Onera, CNRS, Centrale Lille, Arts et Mtiens Paris Tech, FRE 2017 - LMFL - Laboratoire de Mecanique des Fluides de Lille Kamp de Friet, — The log profile overlap solutions for turbulent channel have been complemented recently by solutions for the dissipation\(^1\), \(\varepsilon\) and the kinetic energy\(^2\), \(\langle q^2\rangle/2\). The dissipation varies as \(1/y^\alpha\), while the turbulence kinetic energy varies logarithmically. The Reynolds shear stress is nearly constant. We show from similar arguments that the transport moments, \(\langle \theta T \rangle / \langle q^2 \rangle/2 \) also vary logarithmically. So all the terms in the kinetic energy balance in overlap region, \(\langle \theta T \rangle / \langle q^2 \rangle/2 = \varepsilon \), vary inversely with \(y^\alpha\). Boundary layer results are the power-law equivalents, but indistinguishable. Both are shown to be consistent with recent experimental and DNS data. This presents a problem for the usual eddy viscosity models for this region, \(\nu_t \propto \langle q^2 \rangle/2 \varepsilon\), since both cannot be true. References: 1) Wosnik, M. \textit{et al.} (2000) JFM 421, 115; 2) Hultmark, M. (2012) JFM 707, 575.

Revealing Some Roots to Our Uncertainty Regarding Values of Von Kármán Constants. HASSAN NAGIB, IIT, Chicago, USA, LUCIA MASCOTELLI, GABRIELE BELLANI, ALESSANDRO TALAMELI, Università di Bologna, Forlì, Italy — Three different fits of logarithmic dependence of centerline velocity, \(U^{+}_{CL}\), in pipe flow at CICLOPE, with nearly equal representation of experimental data over range 8,000 < \(Re_{\nu}\) < 40,000 have been used to generate synthetic data that are densely spaced with Reynolds number. A higher order term proportional to inverse of \(Re_{\nu}\) was incorporated into one. Two approaches to uncertainty analysis of three sets of synthetic data were used to study dependence of uncertainty of extracted values of \(k_{CL}\): Random Sampling of Full Range of Uncertainty and All Possible Perturbations of Extreme Uncertainties. Role of low Reynolds number data and accuracy of pressure transducer used to measure pressure gradient along pipe were examined. Results reveal following mean values and uncertainties from measurements by the multiple-transducer pressure scanner that was used: \(k_{CL} = 0.44 \pm 0.006\) for all Reynolds numbers, and \(\kappa_{CL} = 0.45 \pm 0.038\) for data with \(Re_{\nu} > 12,000\). More significantly, we conclude that CICLOPE requires a more accurate pressure scanning method for determining \(dp/dz\) e.g., using a Scanivalve connected to a single more accurate pressure transducer. Such an approach has potential of reducing uncertainty by an order of magnitude.

Turbulent flow field in the viscous sublayer: statistics and structures. SANTOKHUM SANKAR, Dept. of Mechanical Eng & Saint Anthony Falls Lab, Univ of Minnesota, JIARONG HONG, Department of Mechanical Engineering & Saint Anthony Falls Lab, University of Minnesota — The study of wall-bounded turbulence has focused on the logarithmic region above the wall. However, the need to unravel the effects of sublayer roughness, turbulence remains essentially smooth-wall-like, and show how to obtain the position of the virtual origins for the mean velocity and for turbulence from the three slip-length coefficients, and from them the change in drag.

1Supported by NSFC (11452002, 11521091).

1Office of Naval Research, Grant No. N000141612755.
in the symmetric subspace. Periodic orbit and provide strong evidence that the associated homoclinic tangle forms the chaotic repeller that underpins transient turbulence or orbits – as well as continua of connections forming higher-dimensional connecting manifolds. We also compute a homoclinic connection of a

In particular, we find numerous isolated heteroclinic connections between different types of solutions – equilibria, periodic, and quasi-periodic connections between such solutions in a weakly turbulent quasi-two-dimensional Kolmogorov flow that lies in the inversion-symmetric subspace.

F. SCHATZ, ROMAN O. GRIGORIEV, Georgia Institute of Technology, USA — Recent studies suggest that unstable recurrent solutions of the

Flow domain and compared to exact solutions of the Navier-Stokes equation obtained via fully-resolved direct numerical simulation.

η
Taylor-Couette flow with radius ratio Γ = 1

measurements with numerical simulations at the same parameter values and boundary conditions in a small-aspect-ratio (Γ = 1)
hypothesis to obtain laboratory measurements. Here we report evidence for ECS in a 3D turbulent flow by directly comparing experimental

domains that differed from the inflow-outflow boundary conditions of corresponding experimental tests, which relied on the use of Taylor’s

horseshoe which generates chaos and can be evidence to support a theoretical description of the onset of spatially localized transient turbulence – i.e., turbulent puff – observed in pipe flow experiments.

1JRL’s doctoral study is supported by the Japanese Government’s MEXT scholarship (id no. 162100)

8:13AM C11.00002 Self-similar invariant solution in the near-wall region of a turbulent boundary layer at very high Reynolds numbers1, TOBIAS M. SCHNEIDER, SAAJAD AZIMI, EPFL - Swiss Federal Institute of Technology Lausanne — At sufficiently high Reynolds numbers, shear-flow turbulence close to a wall acquires universal properties. When length and velocity are rescaled by appropriate characteristic scales of the turbulent flow and thereby measured in inner units, the statistical properties of the flow become independent of the Reynolds number. We demonstrate the existence of a wall-attached exact invariant solution of the fully nonlinear 3D Navier-Stokes equations for a parallel boundary layer that captures the characteristic self-similar scaling of near-wall turbulent structures. The solution branch can be followed up to Re = 500,000 corresponding to a friction Reynolds number in the millions. Combined theoretical and numerical evidence suggests that the solution is asymptotically self-similar and exactly scales in inner units for Reynolds numbers tending to infinity. Demonstrating the existence of invariant solutions that captures the self-similar scaling properties of turbulence in the near-wall region is a step towards extending the dynamical systems approach to turbulence from the transitional regime to fully developed boundary layers.

1This work is supported by the Swiss National Science Foundation SNSF under grant no. 200021-160088

8:26AM C11.00003 Heteroclinic and Homoclinic Connections in a Kolmogorov-Like Flow1, BALACHANDRA SURI, Institute of Science and Technology, Austria, RAVI KUMAR PALLANTLA, LOGAN KAGEORGE, MICHAEL F. SCHATZ, ROMAN O. GRIGORIEV, Georgia Institute of Technology, USA — Recent studies suggest that unstable recurrent solutions of the Navier-Stokes equation provide new insights into dynamics of turbulent flows. In this study, we compute an extensive network of dynamical connections between such solutions in a weakly turbulent quasi-two-dimensional Kolmogorov flow that lies in the inversion-symmetric subspace. In particular, we find numerous isolated heteroclinic connections between different types of solutions – equilibria, periodic, and quasi-periodic orbits – as well as continua of connections forming higher-dimensional connecting manifolds. We also compute a homoclinic connection of a periodic orbit and provide strong evidence that the associated homoclinic tangle forms the chaotic repeller that underpins transient turbulence in the symmetric subspace.

1National Science Foundation Grants: CMMI-1234436, DMS-1125302, CMMI-1725587. Defense Advanced Research Projects Agency grant HR0011-16-2-0033

8:39AM C11.00004 Unstable Periodic Orbits in Experimental Kolmogorov-Like Flow1, LOGAN KAGEORGE, Georgia Inst of Tech, BALACHANDRA SURI, IST Austria, ROMAN GRIGORIEV, MICHAEL SCHATZ, Georgia Inst of Tech — The geometry of state space for a moderately turbulent flow is shaped by non-chaotic Navier-Stokes solutions known as Exact Coherent Structures (ECS). It has been shown in numerical studies of pipe flow that unstable periodic orbits, one such ECS, guide the evolution of nearby trajectories and form the backbone of the chaotic attractor. However, until now little experimental work has been done to show if periodic orbits are frequently visited in fluid systems and are therefore relevant to the dynamics of the system. We report on numerical work to identify periodic orbits and describe their dynamical relevance in experiments of weakly turbulent quasi-two-dimensional Kolmogorov-like flows.

1This work is supported by DARPA (HR0011-16-2-0033) and NSF (CMMI-1725587)

8:52AM C11.00005 Experimental evidence of exact coherent structures in small-aspect-ratio Taylor-Couette flow1, CHRISTOPHER J. CROWLEY, Georgia Institute of Technology, MICHAEL C. KRYGIER, Sandia National Laboratories, WESLEY TOLER, ROMAN O. GRIGORIEV, MICHAEL F. SCHATZ, Georgia Institute of Technology — Recent work suggests that the dynamics of turbulent wall-bounded flows are guided by unstable solutions to the Navier-Stokes equation that have nontrivial spatial structure and temporally simple dynamics. These solutions, known as exact coherent structures (ECS), are presumed to play a key role in a fundamentally deterministic description of turbulence. Prior work in 3D fluid flows computed ECS in streamwise-periodic domains that differed from the inflow-outflow boundary conditions of corresponding experimental tests, which relied on the use of Taylor’s hypothesis to obtain laboratory measurements. Here we report evidence for ECS in a 3D turbulent flow by directly comparing experimental measurements with numerical simulations at the same parameter values and boundary conditions in a small-aspect-ratio (Γ = 1) turbulent Taylor-Couette flow with radius ratio η = 0.71. To detect an ECS, time-resolved 3D–3C velocity measurements were performed in the entire flow domain and compared to exact solutions of the Navier-Stokes equation obtained via fully-resolved direct numerical simulation.

1Supported by ARO (grants W911NF-15-10471, W911NF-16-10281)
9:05AM C11.00006 The mechanism of long turbulent lifetimes in a low-dimensional model of plane Couette flow. JAMES HITCHEN, ALEXANDER MOROZOV, SUPA, School of Physics and Astronomy, The University of Edinburgh, James Clerk Maxwell Building, Peter Guthrie Tait Road, Edinburgh, EH9 3FD, United Kingdom — Recently, our understanding of the transition to turbulence has significantly changed due to the discovery of exact solutions of the Navier-Stokes equations and the introduction of the self-sustaining process in parallel shear flows. This theory has been very successful in describing the main features of weakly turbulent states, including the metastable nature of turbulence close to the transition and the super-exponential dependence of its lifetime on the Reynolds number. The main strength of this approach is that it allows for a semi-analytical description of the turbulent dynamics in the form of a rather low-dimensional model. Here we systematically develop a novel low-dimensional model that allows us to investigate the origin of the very long turbulent life-times close to the transition. We find that there exists a particular periodic orbit that acts as a porous reflecting barrier between the laminar and turbulent states, and that serves to greatly increase the time before relaminarisation.

9:18AM C11.00007 Heteroclinic Connections as Predictors of Extreme Events in Weakly Turbulent Flow1. JOSHUA PUGHE SANFORD, ROMAN O. GRIGORIEV, Georgia Institute of Technology — Formally, extreme events occur in dynamical systems when an observable varies from its mean by many standard deviations. In practice, this describes events such as earthquakes, rogue waves, seizures, and heart attacks. Due to the catastrophic nature of these events on both an environmental and personal scale, an understanding of how these events occur is crucial. We investigate a two-dimensional Kolmogorov flow, where extreme events correspond to short-lived spikes in global dissipation. Using direct-adjoint iterations, we found a number of heteroclinic connections between unstable recurrent solutions characterized by very different rates in dissipation. We also found that extreme events correspond to the turbulent trajectories shadowing some of these connections in state space. These results suggest that heteroclinic connections may be used to understand and predict extreme events in this and other systems.

1This material is based upon work supported by the National Science Foundation under Grant No. CMMI-1725587 and the Army Research Office under grant No. W911NF-15-10471.

9:31AM C11.00008 Finding unstable periodic orbits with polynomial optimization, with application to a nine-mode model of shear flow1. MAYUR LAKSHMI, GIOVANNI FANTUZZI, Imperial College London, JESUS FERNANDEZ-CABALLERO, University of Warwick, YONGYUN HWANG, SERGEI CHERNYSHENKO, Imperial College London — It was recently suggested (Tobasco et al. Phys. Lett. A, 382, 382–386, (2018)) that trajectories of ODE systems which optimize the infinite-time average of an observable can be localized using sublevel sets of a function that arise when bounding such averages using so-called auxiliary functions. This talk will demonstrate that this idea allows for the computation of extremal unstable periodic orbits (UPOs) for polynomial ODE systems. We first show that polynomial optimization is guaranteed to produce near-optimal auxiliary functions, which are required to localize the extremal UPO accurately. We then show that points inside the relevant sublevel sets can be computed efficiently through direct nonlinear optimization. Such points provide good initial conditions for UPO computations. We illustrate the potential of this new technique of finding UPOs by presenting the results of applying it to a nine-dimensional model of a sinusoidally forced shear flow.

9:44AM C11.00009 The recurrence of flow structures in a low Re wake downstream of two cylinders1. HUIXUAN WU, MEIHUA ZHANG, University of Kansas, ZHONGQUAN ZHENG, Utah State University — The recurrence network method is used to study the evolution of coherent structures in the wake downstream of two cylinders. The upstream cylinder is fixed and the downstream cylinder oscillates in the transverse direction at a fixed frequency. The vortices shed from the upstream cylinder interact with those from the other cylinder, generating a complicated vorticity distribution. Proper orthogonal decomposition is used to extract coherent structures from the vorticity field, but the modes, which are spatial distributions of vorticity, provide limited information about the temporal evolution of the system. In order to analyze the evolution, the time dependent modal coefficients are used to construct a high dimensional phase space. The flow evolution is represented by a trajectory in this space. At the Reynolds number studied in this work, the system keeps visiting the neighborhood of a previous state, though it never exactly repeats itself. This kind of visit can be regarded as recurrence within a certain tolerance. The network manifests a few recurrence patterns, which corresponds to the ways of evolution of the flow field. The result shows that the recurrence network is an effective tool to analyze the temporal evolution of coherent structures.

1University of Kansas General Research Funding

Sunday, November 24, 2019 8:00AM - 10:10AM – Session C12 Vortex Dynamics and Vortex Flow: Wakes 303 - Diana Sher, University of Toronto

8:00AM C12.00001 Dr. forces on a bluff body shedding a 2P wake1. EMAD MASROOR, MARK A. STREMLER, Virginia Tech — We develop analytical expressions for the drag experienced in a uniform flow by a bluff body shedding a 2P wake. Following von Karman’s well-known method for the drag experienced by a fixed cylinder with a typical ‘von Karman street’-type wake, we set up a potential flow past a general bluff body and model its wake as an infinite periodic collection of point vortices with 4 vortices per period. The relative positions of the 4 vortices are calculated by determining the relative equilibrium configurations of 2P wakes. By considering the momentum flux into and out of a rectangular region enclosing the body and cutting through the wake, we determine the drag forces exerted on the cylinder over one period of vortex shedding. The drag coefficients predicted by this method will be compared to those predicted for objects shedding a standard von Karman street, giving insight into the effect of wake type on the drag.

1Support from the National Science Foundation Graduate Fellowship

8:13AM C12.00002 2P or not 2P? , JASON DAHL, ERDEM AKTOSUN, University of Rhode Island — The characterization of modes in the wake of an oscillating cylinder in a free stream has become a common method for describing the spatial variation of vorticity and coherent vortices in these wakes. Arguably, the most commonly observed wake modes come from the classification of Williamson and Roshko (1989) for a cylinder oscillating in the crossflow direction with definitions such as ‘2S’ to describe two single vortices shed per cycle of motion and ‘2P’ to describe two pairs of opposite rotating vortices shed per cycle. When a cylinder is allowed to move in both the in-line and cross-flow direction, however, these simplified classifications of the wake become muddled, as the phasing and pinch-off of vortices can result in intermediate wakes that might fall in between typically observed wake modes, form new complex combinations of vortices, or form strongly three-dimensional wakes with little coherent vortex structure in a 2-D plane. Through a series of forced motion experiments of a circular cylinder undergoing forced in-line and cross-flow motion in a free stream, this complex wake variability is demonstrated, with focus particularly on formation of the ‘2P’ mode, leading to the question: 2P or not 2P?
8:26AM C12.00003 Optimization of mangrove-root inspired arrangements for minimizing wake disturbances. AISHWARYA S. NAIR, AMIRKHOSRO KAZEMI, OSCAR M. CURET, SIDDHARTH VERMA, Florida Atlantic University — Coastal protection measures such as seawalls are vital for reducing the impact of storms and risk of flooding. The root systems of red mangroves, which are a part of certain coastal ecosystems, provide a natural barrier that dissipates wave energy effectively. The replication of such systems could benefit the design of coastal protection infrastructure substantially. In this work, we present simulations of flow across arrays of circular cylinders, as a simplified model of the root network. The objective is to determine the optimal root arrangements that minimize the disturbances in their wake. We couple these simulations with a genetic optimization algorithm, NSGA II, to discover the optimal configurations that generate the least entrainment in their wake. The configurations that produce the highest and lowest entrainments were studied experimentally using Particle Imaging Velocimetry, and a soap film setup. The resulting flow patterns are analyzed to relate the increase or decrease in wake-disturbance to interactions among vortices shed at the individual cylinder level within the arrangements.

8:39AM C12.00004 Vortex Shedding Dynamics for Nonparallel Tandem Cylinders. DIANA SHER, SEAN BLANEY, PHILIPPE LAVOIE, University of Toronto — Nonparallel tandem cylinder geometries are found in many industrial applications. This type of geometry can lead to significant flow induced vibration and noise emissions. However, the flow dynamics is not well understood since this geometry has not received as much attention as isolated or parallel tandem cylinders. This work considers an upstream circular cylinder perpendicular to the flow and a yawed downstream cylinder. Measurements were conducted in an open jet anechoic wind tunnel (AWT) at UTIAS. Yaw angles up to 45° over a range of Re = 40,000-120,000 were tested. The vortex shedding behavior along the span of the cylinders was investigated using surface pressure measurements. Peak patterns observed in the spectra indicate the presence of spanwise-dependent flow cells. A single dominant peak in the frequency spectrum exists where the spacing between the cylinders is narrow, two prominent peaks appear at intermediate spacings, and at locations where the spacing is wide, a single peak is present. There are two distinct shedding frequencies at which these peaks occur for each yaw angle, suggesting the existence of two flow cells. The shedding frequency increases for larger yaw angles independently of Re, with the lower frequency peak exhibiting a larger shift.

8:52AM C12.00005 Wake mode variability for a circular cylinder in a free stream with force sinusoidal in-line and cross-flow motion. ERDEM AKTOSUN, JASON DAHL, University of Rhode Island — Quantitative flow visualization and force measurements were systematically obtained for a circular cylinder undergoing forced sinusoidal in-line and cross-flow motion in a free stream. Variation of the in-line amplitude, cross-flow amplitude, reduced velocity, and phase between in-line and cross-flow motions were made for a fixed Reynolds number of 7620, with a total of 819 experiments for flow visualization and 9555 for force measurements. Visual categorization of the wake shows a wide variability in pattern of vortices shed in the wake, resulting in a wide variability in the resultant forces acting on the cylinder. Although all motions are symmetric in the cross-flow direction, some parameter combinations are observed to produce an asymmetric wake and strong mean lift forces. For a simple demonstration of the complex variation of the wake, one example baseline motion is shown, where small perturbations from the baseline motion are given to demonstrate wake changes as a function of the motion parameters.

9:05AM C12.00006 Vortex-Induced Vibrations of a One-Degree-of-Freedom Cylinder Transitioning from the Inline to the Transverse Degree of Freedom. BRIDGET BENNER, YAHYA MODARRES-SAEDEGH, University of Massachusetts Amherst — Flow-induced oscillations of a cylinder with one degree of freedom in the purely inline direction, and at angles that deviate from the inline direction, are studied experimentally. Experiments are conducted in a recirculating water tunnel using a setup with a low mass-damping coefficient. Force and displacement measurements together with flow visualization of the wake are used to characterize the response of the cylinder over a range of reduced velocities as the single degree of freedom incrementally deviates from the inline direction. It is shown that the two non-zero amplitude regions that are observed at reduced velocities of 1.8 and 2.5 in a purely inline vortex-induced vibration (VIV) response of the cylinder slowly disappear as the angle is increased and the lock-in region, usually observed in a pure crossflow VIV response, appears at higher angles. It is also shown how the observed vortex shedding modes change as the degree of freedom deviates from a pure inline direction.

9:18AM C12.00007 Pulsating Flow Past a Square Cylinder at Low Reynolds Number: Analysis of Vortex Structures. THOMAS FOWLER, IV, FREDDIE WITHERDEN, SHARATH GIRIMAJI, Texas A&M University — Flow past a fixed square cylinder is a canonical problem for investigating vortex-induced vibration and various wake flow physics of interest to several engineering fields. A variant of this problem is that of a pulsating inflow condition. In this work, direct numerical simulations were performed for the case of pulsating flow at Re = 200 over a range of forcing frequencies. As in literature, three regimes are identified: (i) Pre-Lock-in; (ii) Lock-in; and (iii) Post-Lock-in. In Pre-Lock-in, vortex shedding is asymmetric and aperiodic, with the shedding frequency matching that of the uniform case. During Lock-in, vortex shedding remains asymmetric, but becomes distinctly periodic owing to the synchronization of the detachment of the primary and secondary vortices. Here the vortex shedding frequency is determined by the forcing frequency, leading to an increase in the forces experienced by the body. Transitioning to Post-Lock-in, the increasingly strong pulsations lead to symmetric detachment of the primary vortices, disrupting the asymmetric shedding of secondary vortices, and returning to aperiodicity. Spectral analysis then provides further insight regarding the sharp transition into the lock-in regime as opposed to the gradual transition beyond.

9:31AM C12.00008 Effect of Eliminating Trailing Edge Vortices on Thrust Coefficient in a Plunging Flat Plate. AVELINA RAHMAN, PhD Student, DANESH TAFTI, Professor — Plunging motion, characterized by frequency and amplitude is a key component in the kinematics of many flying and swimming organisms. We studied plunging of a flat plate with a broad range of reduced frequencies 0.25 ≤ k ≤ 16 and plunge amplitudes 0.03125 ≤ h ≤ 0.1 giving plunge velocities of 0.25 ≤ k h ≤ 5 at Re=100. This study observed that, unlike previous investigations for small plunge amplitudes, thrust does not increase monotonically with kh but reaches a maximum and then decreases. It is shown that Leading Edge Vortices (LEV) are responsible for thrust production whereas Trailing Edge Vortices (TEV) induce drag on the plate. At higher kh, vortex induced velocities dominate the flow with strong nonlinear vortex-vortex interactions (VVI). Three main VVI mechanisms are identified: in two of them TEV adversely affect thrust production. It is shown that by introducing a splitter plate that eliminates the formation of TEVs, the thrust coefficient (C_T) increases monotonically with kh. A parametrization of the thrust coefficient is done with frequency (k) and amplitude (h) C_T = A k^{1.4} h - B where A and B are constants, with a R^2 =0.96 for the proposed equation. Additionally, a scaling analysis is done between C_T and circulation to see the effect of eliminating TEV on LEV dynamics.
9:44AM C12.00009 Sparse sensor placement for machine learning classification of pitching and plunging plates, JONATHAN TU, NSWC Carderock — We consider the task of using downstream measurements to characterize the upstream motion of a rigid flat plate, which can be thought of as a simplified model of a swimming fish. Specifically, we numerically simulate pitching and plunging plates at a Reynolds number of 100, then apply Linear Discriminant Analysis (LDA) to distinguish between the two upstream motions. To reduce the dimension of the feature space, we apply LDA to projection coefficients of the flow field obtained using proper orthogonal decomposition (POD). However, at inference time this still requires collecting full flow field data, in order to perform the POD projection. To avoid having to collect full flow fields at all, we also use the Sparse Sensor Placement Optimization for Classification (SSPOC) algorithm to find a small number of point sensors that provide the same information as a POD projection. We implement both the original algorithm and a new extension of SSPOC for vector-valued measurements. Whereas the standard SSPOC algorithm might choose certain sensor locations for the \( u \) velocity and others for the \( v \) velocity, our vector SSPOC algorithm places sensors for \( u \) and \( v \) at the same locations. Classification using either SSPOC variant achieves similar accuracy to that using full flow field data.

9:57AM C12.00010 Dynamics of wing tip vortices in the near and far wake, MARIE COULIOU, KEVIN AZIM, JEAN-CLAUDE MONNIER, VINCENT BRION, DAAA, ONERA, ONERA TEAM — Wing tip vortices are a major challenge for civil aviation, both regarding air transport management (efficiency, safety) and environmental impact (induced drag, condensation trails). In this work, an experimental investigation on the development of cooperative instabilities of vortex wakes in the near and far field of a finite wing is carried out. The experiment takes place in a 22 m long hydrodynamic towing tank capable of towing speeds up to 5 m/s and enabling PIV measurements up to 200 spans of wake development. PIV data provides an extensive characterization of the vortex evolution. The effect of a perturbation placed along the span of the wing, following recent theoretical results on the control of vortex wake dynamics, is particularly discussed.

Sunday, November 24, 2019 8:00AM - 9:57AM – Session C13 Convection and Buoyancy-driven Flows: Environmental Flows 304 - Nigel Kaye, Clemson University

8:00AM C13.00001 Flow structures and kinetic-potential energy exchange in forced rotating stratified turbulence, TIANYI LI, Peking University, MINPING WAN, JIANCHUN WANG, SHIYI CHEN, Southern University of Science and Technology — We investigate the long-time evolution of flow structures and the kinetic-potential energy exchange \( e_{K\text{to}P} \) in rotating stratified turbulence. Numerical simulations of forced homogeneous rotating stratified turbulence with different Froude numbers are performed. In the presence of stratification, with two box scale structures with opposite signs of vertical vorticity formed at later times, there are numerous small vortices spreading in the flow. Cyclonic vortices, though being less numerous than anticyclonic vortices, grow faster, thus forming the box-scale structure earlier. Moreover, the intense area of \( e_{K\text{to}P} \) is associated with the cyclonic structures. We also find a relation between vertical vorticity and the distribution of the density in the vortices during the evolution of turbulence.

8:13AM C13.00002 Prandtl Number Dependence of Stratified Turbulence, JESSE LEGASPI, MICHAEL L. WAITE, University of Waterloo — Stratified turbulence is affected by buoyancy forces that suppress vertical motion, resulting in horizontal layers of quasi-two-dimensional vortices. The Prandtl number \( Pr \) (or Schmidt number) quantifies the relative strengths of viscosity and buoyancy diffusivity which damp small-scale fluctuations at different microscales. Direct numerical simulations (DNS) must resolve the smallest features, requiring high resolution for large \( Pr \) (e.g. \( Pr = 7 \) in heat-stratified water and 700 in salt-stratified water). To reduce this computational demand \( Pr = 1 \) is often assumed, possibly introducing discrepancies between DNS and real geophysical flows. In this work, DNS of homogeneous forced stratified turbulence with \( Pr = 0.7, 1, 2, 4, \) and 8 are performed for varying stratification strength. Energy spectra, buoyancy flux spectra, spectral energy flux, and physical space fields are compared for scale-specific \( Pr \)-sensitivity. Intermediate and large scale \( Pr \)-dependence was found in addition to expected small-scale sensitivity. Based on our results, the \( Pr = 1 \) assumption is not realistic for DNS of \( Pr > 1 \) stratified turbulence: the effects at intermediate, and in some cases, large horizontal scales must be considered, though the computational demand can be prohibitive.

8:26AM C13.00003 Examination of temperature spectra in stably stratified boundary layers measured using nano-scale probe, FIONA SPENCER, University of Washington, TYLER VAN BUREN, University of Delaware, ALEXANDER J SMITS, Princeton University, OWEN WILLIAMS, University of Washington — Thermally stable turbulent boundary layers are prevalent in the polar regions and nocturnal atmospheric surface layer but measurements are challenged by changing conditions and small fluxes. Here, we examine the influence of increasing stratification on the spectrum of temperature fluctuations in a laboratory-scale boundary layer over a rough surface. A nanoscale cold-wire (T-NSTAP) is employed to significantly increase frequency response and resolution compared to conventional cold-wires. This method is used to examine boundary layer conditions from near-neutral, through to the collapse of turbulence. This novel dataset allows examination of changes to temperature spectra, their energy cascade and wall-normal locations of maximum spectral energy in both inner and semi-local coordinates. These results when compared to atmospheric data, provide insights into the separation of Reynolds and Richardson number influences.
8:39AM C13.00004 Dynamics of subsiding shells in actively growing clouds with vertical updrafts, VISHNU NAIR, Imperial College London — The dynamics of a subsiding shell at the edges of actively growing shallow cumulus clouds with updrafts is analyzed using direct numerical simulations with grid sizes up to 3072 x 1536 x 1536. The actively growing clouds have a fixed in-cloud buoyancy and velocity. Turbulent mixing and evaporative cooling at the cloud edges generate a subsiding shell which grows with time. A self-similarity analysis reveals that contrary to classical self-similar flows, the turbulent kinetic energy budget terms and the velocity moments scale according to the buoyancy and not with the mean velocity. The shell accelerates ballistically with a magnitude dependent on the saturation value of the buoyancy of the cloud-environment. Tint influence. In this regime, the shell is buoyancy-driven and independent of the in-cloud velocity. The shell thickness and the velocity continue to grow indefinitely and could possibly be limited only by the lifetime of the cloud or thermal. The entrainment coefficient is observed to be a function only of the initial state of the cloud and the environment. This coefficient is linked to the fractional entrainment rate used in cumulus parameterization schemes for large scale models and is shown to be of the same order of magnitude.

8:52AM C13.00005 Impact of ambient stratification on gravity currents propagating over a submerged canopy, JIAN ZHOU, University of California, Berkeley, S. KARAN VENAYAGAMOORTHY, Colorado State University — The dynamics of lock-exchange gravity currents propagating over a submerged canopy in a linearly stratified environment is investigated using highly resolved three-dimensional large-eddy simulations. The canopy is composed of a bottom-mounted array of square cylinders. It is highlighted that the structure and propagation of the gravity currents show remarkable variation across the parameter space of canopy density and ambient stratification. For sparse canopies, the gravity current propagates through the canopy interior, and the reduction in front speed is less sensitive to the canopy drag under stronger ambient stratification where there is less wake-induced buoyancy loss of the current head. For denser canopies, the transition to over-flow on the canopy top and the accompanied recovery of front speed are found to occur earlier in canopy density as the ambient stratification becomes stronger, which is the outcome of three mechanisms: (i) weaker canopy drag due to the altered propagation pathway; (ii) less convective mixing of the over-head with the underlying lighter fluid; and (iii) more regain of effective horizontal buoyancy gradient as the current is lifted up. The latter two mechanisms occur due to the background stratification.

9:05AM C13.00006 Computational modeling of dense gas flushing from urban canyons, RASNA SHARMIN, NIGEL KAYE, Clemson University — Results are presented from a series of CFD simulations of the flushing of a dense pollutant from a model urban canyon. The simulations are run using both RANS and LES simulations for the flushing from a square canopy formed between two square prismatic buildings. Results are presented for the flushing rate for a range of Richardson numbers with higher Richardson number flows exhibiting lower flushing rates. The influence of the upstream surface roughness, formed by imposing a saw-tooth geometry of various aspect ratios on the flow floor, elucidate the role of the boundary layer structure on the flow over the canyon and resulting flushing rate. The computational results are compared to prior experimental results for the same geometry with the goal of further elucidating the flow structure under more controlled conditions.

9:18AM C13.00007 Coaxial moist plumes in stationary and windy environments1, SHUO LI, MORRIS FLYNN, Department of Mechanical Engineering, University of Alberta, Edmonton, AB T6G 1H9 — We report on the dynamics of coaxial plumes in both stationary and windy ambient environments. The coaxial plumes consist of hot, humid air as the inner, circular plume, and warm, less humid air as the outer, annular plume. For a stationary ambient, a double plume model is proposed using a three-way entrainment formulation that involves three undetermined entrainment coefficients. Two body force formulations are discussed, which regard either the ambient or the outer plume as the reference fluid for the inner plume. Meanwhile, a planar laser-induced fluorescence (PLIF) technique is employed to visualize the flow and quantify the scalar concentration of either the inner or outer plumes. The PLIF experiment reveals different near-field entrainment behaviors for cases wherein the inner plume is more/less buoyant than the outer plume. Moreover, the optimal values for the entrainment coefficients are determined by a pixel-by-pixel comparison of the scalar concentration between theory and experiment. For a windy ambient, theoretical results show that wind tends to rapidly dilute the heat and moisture transferred from the inner to the outer plume.

1Natural Sciences and Engineering Research of Canada (NSERC), International Cooling Tower Inc. and the China Scholarship Council (CSC)

9:31AM C13.00008 Large-scale and Small-scale Turbulent Structures in a Stably Stratified Shear Layer1, TOMOAKI WATANABE, Nagoya University, JAMES J. RILEY, University of Washington, KOJI NAGATA, Nagoya University, KEIICHI MASUDA, RYO Q NISHI, JAMSTEC — Title: Large-scale and Small-scale Turbulent Structures in a Stably Stratified Shear Layer Turbulent structures in a stably stratified shear layer are investigated with direct numerical simulations (DNS). Initial mean streamwise velocity and density are given by hyperbolic tangent functions and, therefore, shear and stratification are localized in a thin layer with an initial thickness of \( h_0 \). The DNS uses a very large computational domain in horizontal directions. The Reynolds number and Richardson number based on \( h_0 \) and the density and velocity differences across the shear layer are 2000 and 0.06, respectively. Turbulence develops from the initial conditions, grows, and then it rapidly decays with time. Flow visualization employing the second invariant of the velocity gradient tensor confirms that a large number of hairpin vortices appear near the edge of the shear layer at late time in the simulation. Very elongated flow structures are also found for the streamwise velocity visualized on a horizontal plane, where the length of these long structures is about 10 times larger than the shear layer thickness. It is also shown that the length scales associated with the hairpin vortices and the elongated flow structures make large contributions to turbulent kinetic energy.

1The authors acknowledge JAMSTEC and Nagoya University for offering their computational resources. This work was supported by JSPS KAKENHI (Nos. 18K13682 and 18H01367).

9:44AM C13.00009 On Correlations of Lagrangian and Eulerian Accelerations in Turbulent Stratified Shear Flows, FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Aix-Marseille University — The correlations of Lagrangian and Eulerian accelerations of homogeneous turbulence with uniform shear and stratification are investigated using data from direct numerical simulations. In order to vary the relative importance of shear and stratification, a Richardson number range from \( R_i = 0 \) corresponding to unstratified shear flow, to \( R_i = 10 \), corresponding to strongly stratified shear flow, is considered. The correlations between Lagrangian and Eulerian accelerations are observed to increase with increasing Richardson number. Using a wavelet-based scale decomposition of the acceleration signals at different scales, further investigated. It was found that the Lagrangian and Eulerian accelerations are strongly correlated at large, energy-containing scales of motion. However, the correlations decrease with decreasing scale of the turbulent motion and the accelerations are mainly decorrelated at small, dissipative scales of motion. In addition, the correlations of Lagrangian and Eulerian time-rates of change of fluctuating density are also considered. Again, stronger correlations are obtained at larger Richardson numbers and at larger scales of the turbulent motion.
A comparative experimental and numerical study of rotating Rayleigh-Bénard convection in a cylindrical cell

We acknowledge funding from the H2020 European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant number 678634).

Large Scale Vortices in the Rotating Rayleigh-Bénard setup with no-slip boundaries

We acknowledge funding from the H2020 European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant number 678634).

Probing Regimes and Transitions in Rapidly Rotating Rayleigh-Bénard Convection

We acknowledge funding from the H2020 European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant number 678634).

Effects of Lateral Confinement on Rapidly Rotating Rayleigh-Bénard Convection

We acknowledge funding from the H2020 European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant number 678634).
8:52AM C14.00005 Heat transport by rotating Rayleigh-Bénard convection in cylindrical cells with various aspect ratios1. JIN-QIANG ZHONG, HAO-YUAN LU, JUN-QIANG SHI, Tongji University — Rotating convection has been of interest for decades, yet there exists no generally accepted scaling law for heat transfer behavior in the geostrophic turbulence regime. We present high-precision measurements of the Nusselt number Nu as functions of the Rayleigh number Ra and the Ekman number Ek using cylindrical cells with various aspect ratio Γ. For a given Γ data for Nu(Ra(Ek)) in the geostrophic regime can be represented through one single power function Nu=(Ra/Rac)γ, where Rac=8.7Ek−4/3 is the critical Ra for the onset of convection. However, our experimental and numerical results reveal that the exponent γ increases steeply with increasing Γ, leading to various parameter scaling for the transition towards the geostrophic regime. The present study may provide hints to reconcile previous results of the heat-transport scaling relationship in geostrophic turbulence.

1This work was supported by NSFC-Grant 11572230, 11772235 and 11561161004.

9:05AM C14.00006 Rigorous and numerical upper bounds on heat transport in rapidly rotating Rayleigh-Bénard convection. JARED WHITEHEAD, BENJAMIN PACHEV, Brigham Young University — We present recent rigorous and numerical upper bounds on the heat transport for rapidly rotating Rayleigh-Bénard convection using the asymptotically derived non-hydrostatic quasi-geostrophic equations. We also discuss the challenges inherent to developing upper bounds in the presence of rapid rotation, and propose some novel approaches to the same.

9:18AM C14.00007 Centrifugal Buoyancy and Non-Boussinesq Heat Transfer in Rotating Rayleigh-Bénard Convection1. JONATHAN CHENG, University of Rochester, SIETZE OOSTVEEN, MATTEO MADONIA, RUDIE KÜNNEN, Eindhoven University of Technology — In laboratory experiments of rotating Rayleigh-Bénard convection, conditions differ from the idealized problem in several significant ways. Important among these differences are the presence of centrifugal buoyancy — which causes colder fluid to be driven radially outward and is characterized by the Froude number (Fr) — and non-Boussinesq effects — where variations in the fluid properties break the vertical symmetry of the temperature field. Here, we present a suite of rotating convection simulations under fixed heat flux and Coriolis influence, but with varying centrifugal forcing. The heat transfer is suppressed as Fr increases, with the transition occurring in agreement with the force balance arguments of Horn & Aurnou, Phys. Rev. Lett. 120:204502, 2018. We compare the influence of centrifugation with that of non-Boussinesq effects by examining sidewall temperature gradients in the TROCONVEX rotating convection lab setup. At the sidewall, we expect these two effects to compete against one another. When the Rayleigh number is increased we observe the expected positive shift in the midplane temperature (Ahlers et al., J. Fluid Mech. 596:409-445, 2006), and when Fr is increased, the positive shift is present but reduced.

1This project is funded by the European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant agreement no 678634).

9:31AM C14.00008 Boundary Zonal Flow (BZF) in rotating turbulent convection. EBERHARD BODENSCHATZ, XUAN ZHANG, Max Planck Institute for Dynamics and Self-Organization, DENNIS VAN GILS, University of Twente, SUSANNE HORN, Coventry University, MARCEL WEID, LUKAS ZWIRNER, Max Planck Institute for Dynamics and Self-Organization, GUENTER AHLERS, University of California at Santa Barbara, ROBERT ECKE, Los Alamos National Laboratory, STEPHAN WEISS, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — We report the discovery and overall properties of a wall-localized mode of turbulent, rotating Rayleigh-Bénard convection, denoted as a boundary zonal flow (BZF). The BZF is characterized by the Froude number (Fr) and non-Boussinesq effects — where variations in the fluid properties break the vertical symmetry of the temperature field. Here, we present a suite of rotating convection simulations under fixed heat flux and Coriolis influence, but with varying centrifugal forcing. The heat transfer is suppressed as Fr increases, with the transition occurring in agreement with the force balance arguments of Horn & Aurnou, Phys. Rev. Lett. 120:204502, 2018. We compare the influence of centrifugation with that of non-Boussinesq effects by examining sidewall temperature gradients in the TROCONVEX rotating convection lab setup. At the sidewall, we expect these two effects to compete against one another. When the Rayleigh number is increased we observe the expected positive shift in the midplane temperature (Ahlers et al., J. Fluid Mech. 596:409-445, 2006), and when Fr is increased, the positive shift is present but reduced.

1The work is supported by the Max Planck University of Twente Center for Complex Fluid Dynamics, Los Alamos National Laboratory (the LDRD program), Deutsche Forschungsgemeinschaft (SFB 963, SPP 1881, Sh405/4-1, Sh405/4-2, Sh405/7-1, Sh405/8-1, Ho5890/1-1, We5011/3-1).

9:44AM C14.00009 Properties of the Boundary Zonal Flow (BZF) in rapidly rotating turbulent convection1. XUAN ZHANG, LUKAS ZWIRNER, Max Planck Institute for Dynamics and Self-Organization, SUSANNE HORN, Coventry University, DENNIS VAN GILS, University of Twente, MARCEL WEID, Max Planck Institute for Dynamics and Self-Organization, GUENTER AHLERS, University of California at Santa Barbara, STEPHAN WEISS, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization, ROBERT ECKE, Los Alamos National Laboratory, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — Properties of the boundary zonal flow (BZF) in rapidly rotating Rayleigh-Bénard convection are investigated numerically. Based on direct numerical simulations (DNS) using the finite-volume code GOLDFISH, we analyze in detail the BZF and, in particular, the traveling-mode structures and their dependencies on Ekman number. We also investigate the role of the BZF in the global heat transport in the system as well as the influence of the Prandtl number and the aspect ratio of the cylindrical convection cell on the BZF properties.

1The work is supported by Max Planck University Twente Center for Complex Fluid Dynamics, Los Alamos National Laboratory (LDRD program), Deutsche Forschungsgemeinschaft (SFB963, SPP1881, Sh405/4-1, Sh405/4-2, Sh405/7-1, Sh405/8-1, Ho5890/1-1, We5011/3-1).
9:57AM C14.00010 Experimental investigation of the Boundary Zonal Flow (BZF) in rotating turbulent convection1, STEPHEN WEISS, Max Planck Institute f. Dynamics and Self-Organisation, DENNIS VAN GILS, University Twente, MARCEL WEDI, XUAN ZHANG, Max Planck Institute f. Dynamics and Self-Organisation, SUSANNE HORN, Coventry University, LUKAS ZWIRNER, Max Planck Institute f. Dynamics and Self-Organisation, ROBERT ECKE, Los Alamos National Laboratory, OLGA SHISHKINA, Max Planck Institute f. Dynamics and Self-Organisation, GUENTER AHLERS, University of California, Santa Barbara, EBERHARD BODENSCHATZ, Max Planck Institute f. Dynamics and Self-Organisation, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH (ICTR) COLLABORATION — We report on measurements in rotating turbulent Rayleigh-Bénard convection, in a 2.20 m high cylindrical cell of aspect ratio between its diameter and height of $\Gamma = 1/2$. The working fluids are nitrogen and pressurized (up to 19 bar) sulfur hexafluoride ($\text{SF}_6$). We cover a large Rayleigh number range of $5 \times 10^9 \leq Ra \leq 5 \times 10^{12}$ at Prandtl numbers in the range $0.74 \leq Pr \leq 0.96$. Using thermal probes close to the cylindrical sidewalls we measure characteristic properties of the recently found boundary zonal flow (BZF) as a function of $Ra$ and the rotation rate, i.e., the inverse Rossby number ($1/Ro$). We also discuss in our talk the influence of the BZF on the heat transport.

The work is supported by Max Planck University Twente Center for Complex Fluid Dynamics, Los Alamos National Laboratory (LDRD program), Deutsche Forschungsgemeinschaft (SFB903, SPP1881, Sh405/4-1, Sh405/4-2, Sh405/7-1, Sh405/6-1, Ho5880/1-1, Wi6011/3-1).

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C15 Turbulence: Mixing 310 - Nathanael Machicoane, University of Washington

8:00AM C15.00001 Scalar power spectra and turbulent length scales in high-Schmidt-number scalar fields, MOHAMMAD MOHAGHAR, Georgia Institute of Technology, LAKSHMI P DASI, The Ohio State University, DONALD R WEBSTER, Georgia Institute of Technology — This experimental study has investigated effects of Reynolds number ($5000 \leq Re \leq 20,000$) and initial release diameter ($2.2\text{mm} \leq D \leq 9.4\text{mm}$) on scalar power spectra and turbulent length scales of high-Schmidt-number passive scalar fields resulting from an iso-kinetic release in a turbulent boundary layer. The turbulence analysis is based on 12,000 scalar fields collected using the PLIF technique for each case at 6 locations downstream. Although the spectral slope at intermediate scales is found to increase to an asymptotic value higher than $-5/3$ farther downstream, there is an increase in spectral slope from approximately $-1.5$ for $Re = 5000$ to roughly $-1.2$ for $Re = 20,000$ while fixing the release diameter at 4.7 mm. A similar trend is observed for the effect of nozzle diameter on spectral slope, as it increases from almost $-1.5$ to $-1.2$ when the nozzle diameter changes from 9.4 mm to 2.2 mm while fixing $Re = 10,000$. The scalar integral scale and scalar Taylor microscales are calculated directly from the scalar fields using the correlation function. It was found that the Taylor microscale decreases and the integral scale increases to an asymptotic value respectively, farther downstream. This indicates a larger range of scales exists as flow becomes more turbulent.

8:13AM C15.00002 Frozen waves in turbulent mixing layers, BENOIT-JOSEPH GREA, CEA — Strong time-periodic accelerations applied tangentially to an infinite horizontal plane layer between two miscible fluids trigger a parametric instability leading to remarkable saw-tooth patterns known as frozen waves. The resulting turbulent mixing zones grow in time and then saturate when the resonance conditions of internal gravity waves are no longer fulfilled. The Floquet analysis of a model equation and direct numerical simulations reveal that the final mixing zone sizes evolve as the square of the forcing amplitude while, by contrast, the horizontal wavelengths grow nearly linearly. This explains why frozen waves sharpen with increasingly intense horizontal oscillations. It also suggests that an horizontal forcing mixes more efficiently fluids than a vertical one at large forcing accelerations.

8:26AM C15.00003 Complex network-based Lagrangian analysis of turbulent mixing in channel flows at different Reynolds numbers, GIOVANNI IACOBELLO, STEFANIA SCARSOGLIO, Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Turin, Italy, HANS KUERTEN, Department of Mechanical Engineering, Eindhoven University of Technology, Eindhoven, The Netherlands, LUCA RIDOLFI, Department of Environmental, Land and Infrastructure Engineering, Politecnico di Torino, Turin, Italy — Turbulent mixing is a fundamental issue in many applications, from natural phenomena to industry. In the Lagrangian viewpoint, typical investigations involve the temporal evolution of pairwise mean-square separation or multi-particle shape evolution. In this work, recent advances in complex networks are exploited to study turbulent mixing in a Lagrangian framework. The dynamics of a set of fluid particles is geometricized in a time-dependent complex network, in which nodes correspond to groups of particles and link activation is based on particle spatial proximity. A turbulent channel flow setup is considered at different Reynolds numbers, with the aim to highlight the relative intensity of advection and mixing. Specifically, accurate direct numerical simulations of turbulent channel flows are performed, with Reynolds number between 180 and 950. By doing so, the network-based approach is able to capture the temporal development of particle dynamics due to the turbulent motion, as well as transient and long-term features of wall-normal turbulent mixing. Based on present findings, Lagrangian-based networks can pave the way for a systematic network-based investigation of turbulent mixing, thus extending the level of information of classical statistics.

8:39AM C15.00004 Alignment Analysis of Passive Scalar Mixing in Shock Turbulence Interaction, XIANGYU GAO, IVAN BERMEJO-MORENO, University of Southern California, JOHAN LARSSON, University of Maryland — The transport of passive scalar fields by an initially isotropic turbulent flow passing through a nominally planar shock wave is investigated via shock-capturing DNS for different Mach numbers (1.5 to 5), turbulence Mach numbers (0.1 to 0.4), and Taylor microscale Reynolds numbers (40, 70), with unitary Schmidt number, including both wrinkled- and broken-shock regimes. The effects of the shock-turbulence interaction on alignments between the strain-rate eigenvectors and vorticity, scalar gradient, and the mean streamwise direction are studied, aided by a novel barycentric map representation. Across the shock, the scalar gradient shows an increased alignment with the most extensive eigenvector of strain rate at the expense of a reduced alignment with the most compressive eigenvector (which still dominates) and with the intermediate eigenvector (which becomes nearly perpendicular). This trend is more obvious with larger Mach number and smaller Taylor microscale Reynolds number. Also, across the shock, the most probable alignment between the passive scalar gradient and the eigenvectors of the strain-rate tensor is found to converge towards the alignment that provides the largest scalar dissipation, which is correlated with the enhanced scalar mixing observed downstream of the shock.
A Fractional-Order Non-Fickian Transport Model For Scalar Turbulence

The influence of propagation angle on vortex ring-induced mixing

The largest Schmidt number ratio considered is 64, occurring between two scalars of $Sc = 4$ and 256, whose fluctuations are produced by velocity fluctuations acting upon a uniform mean scalar gradient. Spectral transfer characteristics examined individually for each scalar show robust evidence of a forward cascade, where local transfer by nonlocal interactions modulated by low-wavenumber velocity modes is readily observed at the small scales. Contributions to the coherency spectra from m Molly scalar-non-local velocity-scalar triads were found to produce a net decorrelating effect at small scales which is balanced by the coherent mean gradient forward cascade leading to a stationary state. The scalings of the two-scalar difference spectrum and joint dissipation rate are investigated in detail. The role of differential diffusion in double-diffusive convection is briefly addressed.

Supported by DOE INCITE program at Oak Ridge National Laboratory

This work was supported by the AFOSR Young Investigator Program Award (FA9550-17-1-0150), and partially by MURI/ARO (W911NF-15-1-0562) and by ARO Young Investigator Program Award (W911NF-19-1-0444).

EPSRC
8:00AM C16.00001 An h-adaptive immersed boundary method for simulating fluid-structure interaction. PAN ZHANG, Institute of Applied Physics and Computational Mathematics — Fluid-structure interaction (FSI) problems are encountered in many scientific and engineering applications. As an important calculation model in the field of computational fluid dynamics, immersed boundary method is commonly used to deal with FSI problems. And the method has a wide range of applications. However, for FSI of moving boundary problems for large-scale motion, the moving boundary problem of large-scale motion, the immersed boundary method on uniform grid or non-uniform grid has limitations. This work combines the flexibility of adaptive mesh refinement methods, specifically for uniform Euler mesh. Based on the common characteristics of these two methods, the adaptive mesh refinement method is applied to the immersed boundary method and an adaptive immersed boundary method is presented to adapt to the calculation of the moving boundary problem of large-scale motion.

8:00AM C16.00002 Flow-Structure Interaction Simulation of Parachute using Immerged Boundary Method with Implicit Aerodynamical Load. HANG YU, CARLOS PANTANO, University of Southern California, FEHMI CIRAK, University of Cambridge — A method of flow-structure interaction simulation of the thin structure is presented. The immersed boundary method is used to incorporate the dynamics of the fluid and thin structure. A Cartesian background grid is used for compressible flow simulation and the thin structure is represented by Lagrangian markers immersed in the fluid. Linear operators based on the delta function is used for the interpolation and spreading. The boundary condition is enforced with a singular force \( f \) that is supported on the thin shell \( S \) which serves as boundary representation. Analog to the projection method for incompressible flow, the singular force is regarded as a Lagrangian multiplier to enforce boundary condition. A splitting method is used. The singular force is calculated implicitly by solving a symmetric system, which scales as number of Lagrangian markers. Furthermore, with the interpolation operator acting on the momentum equation, the implicit singular force can be subsequently translated as the aerodynamical load for the thin shell. Therefore, separate calculation of pressure and viscosity load is no longer needed. Lagrangian markers are dynamically removed or added to prevent leakage, unphysical oscillation, or the implicit force equation being underdetermined.

1This material is based on partial work supported by The National Aeronautics and Space Administration (NASA), under Grant Number NNX17AD10G and agent Jessica Powell.

8:26AM C16.00003 An Immersed Boundary Formulation for Fluid-Structure-Interaction in Multi-phase Flows. ELIZABETH GREGORIO, AKASH DHRUV, MEGAN C. LEFTWICH, ELIAS BALARAS, George Washington University — Immersed boundary methods have been widely used to study single-phase flow problems involving complex geometries. However, the extension of existing techniques to multiphase flows is not trivial. When employing direct forcing schemes, the treatment of the level-set function near an immersed boundary is challenging and may result in non-physical mass gain or loss near the interface. In addition, most existing methods adopt a static contact line treatment, which is not suitable for many practical applications. In the present work, we propose a numerical formulation, which uses a moving least squared based forcing for the physical variables and a ghost-cell approach for the level-set function to satisfy a dynamic contact angle boundary condition at the immersed boundary. The method is well suited to model multiphase flow around moving immersed boundaries. A hydrodynamic stress model is also implemented to accurately compute the forces on the body from both the liquid and gas phases. We will establish that the proposed formulation does not introduce mass errors. Comparison with experimental and reference numerical simulations for falling droplets and entry-body problems will demonstrate accuracy and robustness.

8:39AM C16.00004 An immersed boundary method with mesh refinement for turbulent flow simulations. SHAO-CHING HUANG, UCLA — The immersed boundary method has been one of the preferred approaches for direct numerical simulation of turbulence that involves irregular or moving boundaries in the computational domain. The method not only bypasses curvilinear or body-fitted mesh generation but also enables the possibility of using fast solvers in the solution procedure, such as FFT-based methods for Poisson equation, making the method highly efficient. However, the convenience of immersed boundary method can be plagued by lack of resolution in the regions where truncation error is large due to the nature of the underlying structured mesh; increasing the local resolution to resolve sharp gradients usually results in rapid growth of the required total computational resources. To address this problem, we develop continuous forcing of the immersed boundary (IB) method, so that the filter Navier-Stokes equations can be solved on a regular Cartesian grid. Wall shear on the boundary is obtained by solving thin boundary layer equation on an embedded mesh. Subgrid viscosity near the wall is modified to maintain the viscosity flux at the face adjacent to the wall equal to the modeled wall shear stress. Non-physical correlation caused by IB force oscillation is eliminated by applying viscosity buffer under the wall. This method has been validated by simulating turbulent channel and pipe flows at high Reynolds numbers, up to \( Re =10^5 \). Furthermore, flows past a circular cylinder at \( Re =10^4 \) and \( 1.4 \times 10^4 \) have also been simulated. The numerical results are shown to be in good agreements with previous numerical and experiment data.

8:52AM C16.00005 An improved wall model based on immersed boundary method for large eddy simulation of turbulent flow over complex/moving boundaries. MING MA, WEI-XI HUANG, CHUN-XIAO XU, Tsinghua University — A hybrid immersed boundary/wall-model based large eddy simulation method is developed to simulate high Reynolds number turbulent flow with complex/moving boundaries. The no-slip boundary condition is imposed by continuous forcing of the immersed boundary (IB) method, so that the filter Navier-Stokes equations can be solved on a regular Cartesian grid. Wall shear on the boundary is obtained by solving thin boundary layer equation on an embedded mesh. Subgrid viscosity near the wall is modified to maintain the viscosity flux at the face adjacent to the wall equal to the modeled wall shear stress. Non-physical correlation caused by IB force oscillation is eliminated by applying viscosity buffer under the wall. This method has been validated by simulating turbulent channel and pipe flows at high Reynolds numbers, up to \( Re =10^5 \). Furthermore, flows past a circular cylinder at \( Re =10^4 \) and \( 1.4 \times 10^4 \) have also been simulated. The numerical results are shown to be in good agreements with previous numerical and experiment data.

9:05AM C16.00006 A Direct Forcing Immersed Boundary Method for Simulations of Heat and Mass Transport with Neumann Boundary Conditions. JACOB JOHNSTON, JINCHENG LOU, NILS TILTON, Colorado School of Mines — The application of Dirichlet boundary conditions with direct-forcing immersed boundaries is well understood. There is less published work, however, on the application of Neumann conditions, particularly to second-order spatial accuracy in the context of finite volume and projection methods. This issue is important for the simulation of membrane filtration systems in which coupled heat and solute transport occur in the presence of complicated surfaces. Though linear interpolation of the forcing over the grid is sufficient to ensure second-order accuracy of Dirichlet conditions, we find that third-order interpolation is required for the implementation Neumann conditions to second-order accuracy. For semi-implicit simulations, this increases the local stencil such that matrices are no longer banded. We use the method to develop a 2D unsteady finite volume code that simulates heat, mass and momentum transport with the projection method of Choi and Moin (Journal of Computational Physics, 1994). We perform numerical experiments to confirm second-order spatial and temporal accuracy, and then use the method to simulate transport in a membrane filtration system.
9:18 AM C16.00007 A Parallel Dynamic Overset Grid Framework for Incompressible Flow Simulations\footnote{This work is supported by the American Heart Association Grant 13SDG17220022, and the computational resources were partly provided by Texas A&M high performance research computing center (HPRC).}, MOHAMMADALI HEDAYAT, IMAN BORAZJANI, Texas A&M University — The overset grid technique enables the flow solvers to handle unsteady moving grid simulations. However, the task of overset grid assembly in parallel for partitioned grid remains challenging. In this study, a new efficient parallel grid assembly and interpolation framework is developed to perform overset grid assembly for structured meshes in a distributed computing environment. This framework is integrated with a sharp interface curvilinear immersed boundary (CURVIB) flow solver to handle multiple overlapping flow domains. To achieve a good parallel performance several steps are implemented in our framework: 1) using oriented bounding boxes (OBB) instead of axis-aligned bounding boxes; 2) using a walking strategy for donor search; 3) directly integrating grid assembly to the flow solver; 4) efficient vectorized implementation for velocity interpolation; and 5) using a general non-inertial frame of reference flow solver to prevent the recomputation of curvilinear grid. The framework verified and validated against experimental and numerical benchmarks. Our results show a good scalability and accuracy for this new framework. In addition, its capabilities are demonstrated by simulating a school of aquatic swimmers in a diamond shape.

9:31 AM C16.00008 High-Order Ghost-Point Method for Non-Conforming Boundaries\footnote{Laboratory Directed Research and Development (LDRD), Los Alamos National Laboratory}, PRakash Shrestha, Peter Brady, Vitaliy Gryya, Daniel Livescu, Los Alamos National Laboratory — We investigate numerical properties of constrained moving least squares method for numerical implementation of solid boundary conditions (CMLS, an immersed boundary method) by Qu, Shi and Batra (2018) coupled with central finite differences for interior derivatives. This study represents an extension of the original method, which uses first order interpolation / extrapolation for the ghost and image points, as well as dissipative interior discretization. The objectives of the investigation are to determine the suitability of the method for direct numerical simulations of turbulent flows in complex geometries and to find an optimal set of built-in parameters in terms of achieving high order of accuracy and stability of the method for a wide range of canonical test problems. The test problems include a 1-D linear scalar wave equation, for which rigorous stability and conservation properties can be discussed, and 2-D nonlinear tests using Burgers’ equation and the compressible Euler equations with manufactured solutions. Preliminary data indicate that the method can achieve good stability and accuracy properties.

9:44 AM C16.00009 A single-sided direct-forcing diffused immersed boundary method for correct local velocity gradient computation, CHENG PENG, Pennsylvania State University, LIAN-PING WANG, Southern U of Sci and Tech and U of Delaware — Current algorithms of the immersed boundary method (IBM) based on diffused interfaces are not able to correctly calculate the velocity gradients within the diffused solid-fluid interfaces. This is because the non-zero boundary forcing creates a difference in the actual momentum equation solved in IBM from the physical one described by the Navier-Stokes equations with a sharp fluid-solid no-slip interface. A single-sided direct-forcing IBM algorithm is proposed to remove the boundary forcing from the fluid region. The capability of the proposed algorithm in correctly computing velocity gradients within all fluid region is validated in both laminar and turbulent flows. A technique to speed up the convergence of no-slip enforcement via force iteration is also introduced. This technique works for the proposed algorithm and other IBM algorithms.

9:57 AM C16.00010 An Immersed Boundary Method Coupled with a Level Set and Dynamic Overlapping Grids Approaches for Free Surface and Moving Bodies Problems\footnote{Membership Pending}, Riccardo Broglia\footnote{1}, Antonio Posa, Danilo Durante, CNr-lnm — An immersed-boundary (IB) approach was developed within an existing level-set/dynamic overlapping-grids finite-volume solver. An IB strategy is utilized together with curvilinear grids capabilities, keeping cells count under control, a main disadvantage for the classical IB methods using Cartesian grids. Moreover, the IB is identified by an additional level-set function, i.e., a distance function defined at each node of the computational grid, whose zero level represents the fluid/solid interface. One of the main advantages of the proposed approach is the coupling with a dynamic overlapping-grids methodology and a single-phase level-set approach. The former is especially convenient in presence of moving bodies: updating the position of the Lagrangian grid, which discretizes the surface of the body, relative to the Eulerian grid, is not required, since the Eulerian grid attached to a moving IB can follow the body during its motion. The latter is efficiently adopted to handle the presence of an air/water interface. Here the methodology is discussed in detail. Test cases feature stationary and moving bodies as well as complex geometries and free-surface flows. Results from present IB computations are compared with body-fitted solutions and data from literature.

Sunday, November 24, 2019 8:00AM - 10:10AM – Session C17 Focus Session: Recent Advances in Data-driven and Machine Learning Methods for Turbulent Flows I 4c4 - Karthik Duraisamy, University of Michigan

8:00 AM C17.00001 Unsteady Flow Field Predictions Using Multi-level Deep Convolutional Autoencoder Networks\footnote{The authors acknowledge support from the Air Force under the Center of Excellence grant FA9550-17-1-0195, titled “Multi-Fidelity Modeling of Rocket Combustor Dynamics.”}, Jiayang Xu, Karthik Duraisamy, University of Michigan — A machine learning framework is proposed for unsteady flow field predictions. Three levels of deep neural networks are used, with the goal of predicting the future state of the flow for unseen global parameters. A Convolutional autoencoder is used as the top level to encode the high-dimensional data sequence along spatial dimensions into a sequence of latent variables. A temporal convolutional autoencoder serves as the second level, which further encodes the output sequence from the first level along the temporal dimension, and outputs a set of latent variables that fully captures the spatio-temporal evolution of the flow field. A fully connected network is used as the third level to learn the mapping between these latent variables and the global parameters from training data, and predict them for new parameters. For future state predictions, the second level uses temporal convolutional network to predict subsequent steps of the output sequence from the top level. Outputs at the bottom level are decoded to obtain the high-dimensional flow field sequence at unseen global parameters and/or future states.
8:13AM C17.00002 Physics-Constrained Convolutional LSTM Neural Networks for Generative Modeling of Turbulence

ALANSE MOHAN, DANIEL LIVESCU, Los Alamos National Laboratory, MICHAEL CHERTKOV, University of Arizona — High fidelity modeling of turbulence and related physical phenomena is often challenging due to its prohibitive computational costs or the lack of accurate theoretical models. In the recent years, deep learning approaches have shown much promise in modeling of complex systems. A major challenge in deep learning for generative modeling of turbulence is the chaotic, high dimensional and spatio-temporal nature of the data, which makes the learning process ineffective and/or expensive. Previous work by the authors (Mohan et al., 2018) showed the capability of Convolutional LSTM (ConvLSTM) neural networks in modeling high fidelity 3D turbulence. ConvLSTM augments the traditional architecture of an LSTM cell with a convolutional layer to learn spatial features in high dimensional datasets. In this work, we introduce various physical constraints of incompressible turbulent flows into ConvLSTM networks. We demonstrate its efficacy by learning and predicting physically consistent dynamics of a homogeneous isotropic turbulence DNS dataset. Statistical tests are also performed on the predicted turbulence to assess the quality of the physical constraints on the "learned" physics. Finally, we discuss challenges and opportunities with ConvLSTM when enforced with physical constraints, with additional focus on computational scaling of this approach to large datasets.

LANL/DOE LDRD

8:26AM C17.00003 Physics Informed Learning of Lagrangian Turbulence: Velocity Gradient Tensor over Inertial-Radius Geometry

YIFENG TIAN, DANIEL LIVESCU, University of Arizona — The phenomenological model of coarse-grained velocity gradient tensor (VGT) constructed by considering the Lagrangian dynamics of four points, or the tetrad, is extended under the Physics-Informed Machine Learning (PIML) framework. The pressure hessian contribution is re-constructed from the dynamics of Lagrangian tetrad, which provides an improved representation of its magnitude and orientation. The unclosed incoherent small scale fluctuations are modeled using ML techniques trained on Langrangian data from a high-Reynolds number Direct Numerical Simulation (DNS). Certain constraints, such as Galilean invariance, rotational invariance, and zero-pressure work condition, are enforced to implement known physics into the ML model. Then, a comprehensive diagnostic test is performed. Statistics of the flow, as indicated by the joint PDF of second and third invariants of the VGT at different coarse-grained scales, show good agreement with the ground-truth DNS. Some important features regarding the structure of the turbulence are correctly reproduced by the model including skewed distribution of the velocity gradient, vorticity-strain rate alignment and vortex stretching mechanism. The pressure hessian and small-scale contributions to Lagrangian dynamics are also well-captured.

8:39AM C17.00004 Prediction of Aerodynamic Flow Fields Using Spectral Convolutions on Graph Networks

JAMES DUVALL, KARTHIK DURAIKUMAR, YASER AFSHAR, University of Michigan — In this work, spectral methods for performing localized convolutions on graphs are investigated to predict aerodynamic flow fields given the geometry of the surface, and flow configurations therein. Previous work has shown that convolutional neural networks (CNNs) can be used for this purpose. CNNs, however, are restricted to Euclidean domains, and their use requires interpolation from non-regular mesh representations typical of flow solutions on an evenly spaced Cartesian mesh. This represents a loss of information as flow solvers include a clustering of points near boundary layers and other regions of sharp gradients. We pursue graph convolutional networks (GCNs) which operate over non-Euclidean data represented by a graph. GCNs generalize many of the characteristics associated with CNNs. Localized filtering operations are defined in the graph spectral domain, and depend on the graph Laplacian, which is graph structure dependent. Although meshes for different geometries may be spatially distinct, they share spectral characteristics if a binary adjacency matrix is considered. GCNs operating directly on graph representations of spatial flow solver meshes are shown to predict aerodynamic flow fields on unseen airfoil shapes and operating conditions to a good degree of accuracy.

1Graduate Research Assistant, Dept. of Aerospace Engineering.
2Associate Professor, Dept. of Aerospace Engineering.
3Postdoctoral Research Fellow, Dept. of Aerospace Engineering.

8:52AM C17.00005 Potential of using deep neural networks for turbulent-flow predictions

RICARDO VINUESA, PREM A. SRINIVASAN, LUCA GUASTONI, HOSSEIN AZIZPOUR, PHILIPP SCHLATTER, KTH Royal Institute of Technology — The capabilities of deep neural networks to predict temporally evolving turbulent flows are evaluated in this work. To this end, we employ the nine-equation shear flow model by Moehlis et al. (New J. Phys. 6, 56, 2004) as a low-order dynamical representation of near-wall turbulence. We thoroughly tested two different neural networks: the multilayer perceptron (MLP) and the long short-term memory (LSTM) network, and determined the best configurations for flow prediction (i.e., number of layers, number of units per layer, dimension of the input, weight initialization strategy and activation function). Because of its ability to exploit the sequential nature of the data, the LSTM network outperformed the MLP. In particular, relative errors of 0.45% and 2.49% were obtained in mean and fluctuating quantities respectively with the LSTM. Furthermore, this network also led to an excellent representation of the dynamical behavior of the system, characterized by Poincaré maps and Lyapunov exponents. The present results underpin future applications aimed at developing inflow and off-wall boundary conditions for turbulence simulations, and data-driven flow reconstruction of more complex wall-bounded turbulent flows, including channels and developing boundary layers.

1Funding by the Swedish e-Science Research Centre (SeRC) and the Knut and Alice Wallenberg (KAW) Foundation is acknowledged.

9:05AM C17.00006 Turbulence inflow generation using generative adversarial network

JUNHYUK KIM, CHANGMOO LEE, Yonsei University — Using unsupervised learning, we developed an inflow generator that performs better than previous synthetic methods. Direct numerical simulations of turbulent channel flow were carried out at three Reynolds numbers, and then temporally successive flow fields in a cross-sectional (y-z) plane were collected. Using the collected data, we trained a novel model, RNN-GAN, which is composed of a recurrent neural network (RNN) and a generative adversarial network (GAN). Here, RNN represents time-variation of generated flow, while GAN represents a spatial correlation of the flow. Our trained RNN-GAN produces surprising results. First, the generated flow is qualitatively and statistically accurate as compared with DNS. Second, it is possible to generate the flow not only at the trained Reynolds numbers but also at the other Reynolds numbers, although the extrapolated case shows a little deterioration of statistical accuracy. The generated flow is stochastically varying over time, unlike a supervised learning method. Finally, the domain size of the generated flow is extendable. These results indicate that our model provides good inflow generator required for developing channel flow.
9:18AM C17.00007 Physics-informed Spatio-temporal Deep Learning Models, KARTHIK KASHINATH, ADRIAN ALBERT, RUI WANG, MUSTAFA MUSTAFA, Lawrence Berkeley Lab, ROSE YU, Northeastern University — Simulating the spatio-temporal evolution of a complex system over a realistic domain is extremely compute-intensive with current PDE-solvers. Deep learning (DL) shows great promise for augmenting or replacing compute-intensive parts of computational physics models. However, it remains a grand challenge to incorporate physical principles in a systematic manner to the design, training and inference of such models. Physics informed DL aims at defining parameterizations of physical systems into DL models, but existing studies are either limited to purely linear dynamics or purely spatial constraints of physical systems. We study spatiotemporal modeling of velocity fields for a highly nonlinear turbulent flow using various state-of-the-art physics informed DL methods. We benchmark these methods on the task of forecasting velocity fields at different future time horizons, given historic data of different lengths. We find that incorporating prior physics knowledge can not only speed up the training process but improve model performance. Our results show that the Spatiotemporal Generative Networks with an autoregressive U-net as the generator performs the best for varying forecasting horizons.

9:31AM C17.00008 Neural Network Optimization Under Partial Differential Equation Constraints, KARTHIK KASHINATH, Lawrence Berkeley Lab, CHIYU JIANG, U C Berkeley, GAVIN ELI JERGENSEN, MR PRABHAT, Lawrence Berkeley Lab, PHILIP MARCUS, U C Berkeley, PHYSICS-INFORMED DEEP LEARNING COLLABORATION — Enforcing physical constraints to solutions generated by neural networks (NN) remains a challenge, yet it is essential to their accuracy and trustworthiness. We propose a novel differentiable spectral projection layer that efficiently enforces spatial PDE constraints using spectral methods, yet is fully differentiable, facilitating end-to-end training. We train a 3D Conditional Generative Adversarial Network for turbulence superresolution, whilst guaranteeing the spatial constraint of zero divergence. Results show that the model produces realistic flow fields with more accurate flow statistics when trained with hard constraints, compared to soft constrained and unconstrained baselines. We also present a method of applying multiple PDE constraints by modifying the loss function directly. We provide theoretical guarantees of convergence and evaluate the computational complexity of the method. We offer an approximation which trades convergence guarantees for improved speed. Experimentally, we train constrained NN to learn continuous representations of solutions to the linear Helmholtz equation and the nonlinear steady-state Navier-Stokes equation. We show that the model outputs better respect the underlying physics, but note the complexity restricts its application to small NN.

9:44AM C17.00009 Data-driven prediction of a multi-scale Lorenz 96 chaotic system using a hierarchy of deep learning methods: Reservoir computing, ANN, and RNN-LSTM, PEDRAM HASSANZADEH, ASHEESH CHATTOPADHYAY, KRISHNA PALEM, DEVIKA SUBRAMANIAN, Rice University — The performance of three deep learning methods for predicting short-term evolution and reproducing the long-term statistics of a multi-scale spatio-temporal Lorenz 96 system is examined. The methods are: echo state network (a type of reservoir computing, RC-ESN), deep feed-forward artificial neural network (ANN), and recurrent neural network with long short-term memory (RNN-LSTM). This Lorenz system has three tiers of nonlinearly interacting variables representing slow/large-scale (X), intermediate (Y), and fast/small-scale (Z) processes. For training or testing, only X is available; Y and Z are never known/used. It is shown that RC-ESN substantially outperforms ANN and RNN-LSTM for short-term prediction, e.g., accurately forecasting the chaotic trajectories for hundreds of numerical solver’s time steps, equivalent to several Lyapunov timescales. RNN-LSTM and ANN show some prediction skills as well; RNN-LSTM bests ANN. Furthermore, even after losing the trajectory, data predicted by RC-ESN and RNN-LSTM have probability density functions (PDFs) that closely match the true PDF, even at the tails. PDF of the ANN data deviates from the true PDF. Implications, caveats, and applications to data-driven surrogate modeling of complex dynamical systems are discussed.

9:57AM C17.00010 Data-driven super-parametrization using deep learning for large scale turbulent flow in weather/climate modeling, ASHEESH CHATTOPADHYAY, ADAM SUBEL, PEDRAM HASSANZADEH, KRISHNA PALEM, Rice University — Some of the physical processes that play key roles in turbulent systems such as weather/climate systems occur at such small spatial and fast time scales that trying to explicitly solve for them can lead to computationally intractable numerical models. These subgrid-scale processes (denoted by variable Y hereafter), are often parameterized using semi-empirical/physics-based schemes as a function of the large-scale/slow variables (X) that are explicitly solved. Multi-scale numerical models that explicitly solve for X and Y, but at different numerical resolutions, dubbed super-parameterization (SP), has been shown to improve simulations of large-scale turbulence in climate models, but at a large computational cost. More recently, several studies have shown promises of using deep neural networks, trained on data from high resolution climate models, for data-driven parameterization (DDP) of Y as a function of X. Here, we show that Gated Recurrent Units (GRU) can be used for data-driven super-parameterization (DDSP): To solve for X numerically at low resolution and emulate the evolution of Y with a GRU at higher numerical resolutions, at a much lower computational cost, and similar accuracy as that of SP on a multi-scale Lorenz 96 test bed.

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C18 Focus Session: Competing Roles of Surfactants in Free Surface Flows with Hydrodynamic Singularities 400 - Pritish Kamat

8:00AM C18.00001 The Surprising Influence of Marangoni Stress on Near-Singular Dynamics in Breaking Surfactant-covered Liquid Threads, PRITISH KAMAT, Dow Incorporated., OS-MAN BASARAN, Purdue University — The singularity arising during thread breakup is a precursor to drop formation and is accompanied by flows directed away from the breakup location that grow stronger as pinch-off nears. When surfactants are present at interfaces, these outward flows convect surfactants away from the singularity such that the dynamics proceeds in local absence of surfactants. Despite this, it has been observed that even small amounts of surfactants can have a substantive effect on breakup. Experiments show reduced rates of thinning, altered satellite sizes, and formation of fractal structures—microthread cascades—whereas simulations show that thinning filaments can recover and escape pinch-off in the presence of surfactants. Here, we show that evacuation of surfactants from the breakup singularity indirectly leads to the generation of large concentration (surface tension) gradients in the vicinity of but not at the breakup location. The resultant gradients generate Marangoni stresses that are large enough to compete with other forces, and are key to how surfactants influence near-singular dynamics. Here, we use simulations to shed light on two distinct contexts where the competition between Marangoni stresses and other forces perceptibly influences the fate of the breakup singularity.
8:13AM C18.00002 Influence of the surface viscosities on the pinching of a pendant droplet loaded with SDS\textsuperscript{1} , MIGUEL A. HERRADA, Universidad de Sevilla, ETSI, 41092 Sevilla, Spain, ALBERTO PONCE-TORRES, MANUEL RUBIO, JOSE M. MONTANERO, Depto. de Ingeniería Mecánica, Energética y de los Materiales, Universidad de Extremadura, E-06006 Badajoz — The interfacial viscosities of relatively viscous insoluble surfactants are known to affect the dynamics of a liquid thread close to the free surface pinching [1]. Here, we analyze both numerically and experimentally the breakup of a pendant water drop loaded with SDS. We focus on the influence of the monolayer of SDS on the free surface minimum radius \( R \). The results show remarkable agreement between the experiments and numerical simulations for the pure DIW case. When a surfactant is introduced, it creates a monolayer that alters the pinch-off dynamics. If only solute-capillarity and Marangoni convection are considered in the numerical simulations, there is a measurable deviation with respect to the experimental results for \( R_{\text{min}}(\tau) < 5 \) \( \mu \text{m} \). We considered the surface shear and dilatational viscosities in the simulation to reproduce the entire range of experimental data. The value of the shear viscosity is consistent with the upper bound reported in the literature [1]. [1] A. Ponce-Torres, J. M. Montanero, M. A. Herrada, E. J. Vega, and J. M. Vega, Phys. Rev. Lett. 118, 024501 (2017).

8:26AM C18.00003 The role of surface viscous stresses in the liquid thread breakup, HANSOL WEE, Purdue University, PRITISH KAMAT, Dow Incorporated, OSMAN BASARAN, Purdue University — Many industrial processes involving emulsions, foams, and inkjet printing exploit the ability of surfactants to adsorb onto and lower the surface tension of water-air and water-oil interfaces. In addition to lowering surface tension, surfactants may induce Marangoni stresses and cause surface rheological effects. Therefore, the dynamics of free surface flows can be significantly altered by their presence. Although much attention has been paid to date to the influence of Marangoni stresses and solutocapillarity, the effect of surface viscous stresses has been inadequately studied given the difficulty in measuring surface viscosities due to the presence of surfactants. Using a simple model that they vary linearly with concentration, we examine their effect on thread breakup by 1D simulations using the slender-jet approximation. The results obtained with the 1D algorithm are confirmed by direct comparison against predictions made with a 3D but axisymmetric free surface solver.

8:39AM C18.00004 How surfactants influence the drop size in sprays , DANIEL BONN, Institute of Physics, University of Amsterdam — Spraying is a widely used method to produce a liquid sheet that break up into droplets of a certain size distribution. When spraying simple liquids, it is known which experimental parameters determine the droplet size distribution. For many applications however, surfactants are added, producing a hitherto unknown effect on the droplet size distribution. Using two generic types of spraying nozzles, we sprayed solutions of different types of aqueous surfactants and measured the droplet size distribution of the sprays. We find that the breakup of surfactant solutions is similar to that of pure water but results in droplets that are on average smaller. The resulting droplet size distribution can be well described using the predictions for simple liquids provided that we replace the parameter of the equilibrium surface tension with the dynamic surface tension of the surfactant solution at a surface age of 20 ms, which is the characteristic time for destabilization and breakup of a liquid sheet. By rescaling them with the mean droplet size, the droplet size distributions of water and sprays with different concentrations of surfactants all collapse onto a single curve and can be well described using the compound Gamma function found previously for pure liquids.

8:52AM C18.00005 Restoring universality to the pinch-off of a bubble , AMIR PAHLAVAN, HOWARD STONE, Princeton University, GARETH MCKINLEY, RUBEN JUANES, MIT — We observe the formation of bubbles and drops on a daily basis, from dripping faucets to raindrops entraining bubbles on the surface of a lake. The pinch-off of a bubble is an example of the formation of a singularity, exhibiting a characteristic separation of length and time scales. Because of this scale separation, one expects universal dynamics that collapse into self-similar behavior determined by the relative importance of viscous, inertial, and capillary forces. Here, we report on the intriguing observation that confinement makes the pinch-off of a bubble a universal process, as opposed to the unconfined case, where pinch-off is sensitive to the details of the experimental setting. We show that the pinch-off dynamics of a bubble confined in a capillary to a sequence of two distinct self-similar regimes controlled by a balance between viscous and capillary forces. We demonstrate that the early-time self-similar regime restores universality to bubble pinch-off by erasing the system’s memory of the initial conditions. Our observations have implications for immiscible flow phenomena from microfluidics to geophysical flows, where confinement, together with fluid-solid physicochemical interactions, play a key role.

9:05AM C18.00006 Flow statistics of Marangoni contracted sessile drops , STEFAN KARPITSCHKA, OLINKA RAMIREZ, Max Planck Institute for Dynamics and Self Organization, Göttingen, Germany, MICHEL A. HACK, Physics of Fluids Group, University of Twente, Enschede, The Netherlands, WOJCIECH KWIECINSKI, E. STEFAN KOOLJ, Physics of Interfaces Group, University of Twente, Enschede, The Netherlands — A droplet of two miscible liquids should spread over a high-energy surface until complete wetting. However, if one component is more volatile and has a higher surface tension, a quasi-stationary non-vanishing apparent contact angle can be observed. This is caused by the enrichment of the residual component near the contact line and the associated surface tension gradient. A hydrodynamic-evaporative model, using a long-wave approximation for the droplet coupled to diffusion limited evaporation predicts a balance between Marangoni and capillary flows and a power law between the apparent contact angle and the ambient humidity [Karpitschka et al., Langmuir (2017)]. This explanation differs from a recent model, where the low surface tension of a precursor around the droplet is held responsible [Benesiglio et al., Soft Matter (2018)]. A discrimination between possible mechanisms requires experimental resolution of the flow in the drop. We present uPIV measurements and relate them to the apparent shape of the drop, for aqueous solutions of various short chain carbon diols. Depending on the surface activity of the diol, its concentration, and the ambient humidity, we observe different regimes, indicating that multiple mechanisms lead to the observed angles.

9:18AM C18.00007 Marangoni effects enabling interfacial singularities and topological changes in fluid flows , ROUSLAN KRECHETNIKOV, University of Alberta — In this talk I will address two geometric effects of surface tension gradients: formation of interfacial singularities and change in the flow topology compared to the clean interface case. In the first part, I will focus on a necessary condition for the existence of geometric singularities – divergence of curvature at fluid interfaces – in the solutions of fluid dynamic equations. Besides establishing a relation to dynamic singularities – unboundedness of the velocity field – explicit asymptotic solutions of the Navier-Stokes equations are developed as well. Next, using as an example the phenomena of tip-streaming, with the help of asymptotic matching we resolve the associated Marangoni-driven singularities providing explicit asymptotic formulas for the scaling of the emitted droplets. In the second part, using the Landau-Levich problem of dip coating as a paradigm, we demonstrate how Marangoni stresses are capable of changing the flow topology by moving the stagnation point initially residing at the interface in absence of surfactants into the bulk once surfactants are added to the system. This finding not only explains thickening of the deposited film as induced by Marangoni effects, but also illustrates another geometric effect surface tension gradients may cause.
9:31AM C18.00008 Coalescence of surfactant-laden drops in liquid-liquid emulsions
VISHRUT GARG, Air Products and Chemicals, Inc., OSMAN BASARAN, Purdue University — Determining the timescale over which liquid-liquid emulsions separate into their constituents is crucial for many processes, e.g. separating crude oil from brine. This timescale depends on the dynamics of collision and coalescence of liquid drops immersed in a second liquid which can be significantly altered by the presence of surfactants at the liquid-liquid interface. We simulate the approach, collision, and eventual coalescence of two drops immersed in an ambient liquid in the presence of insoluble surfactants where both liquids are incompressible Newtonian fluids. The governing equations are augmented to account for long range van der Waals interactions that become significant as the separation between the drops falls below a few hundred nanometers and solved using a Galerkin finite element based algorithm. In contrast to drops with clean interfaces which coalesce during their first approach, surfactant laden drops are seen to rebound on first approach before coalescing on their subsequent approach. This rebound results in increased drainage times. We examine the physics underlying drop rebound in the presence of surfactants.

9:44AM C18.00009 Coating and Crumpling of Particle-Coated Bubbles in Confined Geometries
SHELLEY ANNA, CHARLES SHARKEY, ZIXIAN CUI, Carnegie Mellon University — We examine bubble flow in a capillary filled with a suspension of surface-active particles. Silica nanoparticles are rendered surface active when mixed with cationic CTAB surfactant, which adsorbs to the silica surface in a glycerol-water mixture. Bubbles are dispersed via a co-flow nozzle at varying bubble lengths and capillary numbers. Fluid film thickness is measured along the length of the bubble. The measurements are compared with bubble flow through surfactant solutions and with predictions from a Bretherton-type model. The adsorbed particle-surfactant complexes form an evolving rigid layer at the trailing end of the bubble. Two critical bubble lengths are observed. Above the first critical length, the bubble contains two distinct film thicknesses. The thickened film at the trailing end arises from the rigid particle layer on the interface. Above the second critical length, the trailing bubble cap crumples and collapses. Crumpling occurs soon after the bubble is dispersed into the capillary, and the critical length varies with bubble velocity. These results allow us to infer rheological and mechanical properties for the interface that are associated with the crumpling phenomenon.

9:57AM C18.00010 Waves on an interface with surfactant
WILLEM VAN DE WATER, Laboratory for Aero and Hydrodynamics, Delft University of Technology, YUKMAN LAU, JERRY WESTERWEEL, Laboratory for Aero and Hydrodynamics Delft University of Technology, DAMIR JURIC, JALEL CHERGUI, Laboratoire d'informatique pour la Mécanique et les Sciences de l'Ingénieur (LIMSI), Centre National de la Recherche Scientifique (CNRS), SEUNGWON SHIN, Department of Mechanical and System Design Engineering, Hongik University, Korea — The presence of surfactant on an interface between two immiscible fluids can dramatically change the interfacial tension. The question is whether this is still so when the interface is rippled through waves which redistribute surfactants. We study Faraday waves on an oil-water interface and use benchmark surfactants with increasing concentration that reaches into the realm of ultralow interfacial tension ($\sigma = O(10^{-10}) \text{N/m}$). In the experiments we measure the wavelength, wave height and threshold acceleration amplitude and compare them to a Floquet analysis. Surprisingly, the dispersion of capillary waves (frequency 20 Hz) points to a much stiffer interface at surfactant concentrations where it should be ultralow. We hypothesize the key role of surfactant dynamics. This is supported by numerical simulations of Faraday waves in the presence of surfactant gradients.


Sunday, November 24, 2019 8:00AM - 10:10AM
Session C19 CFD: High-order and Shock-capturing Methods
401 - Bryce Campbell, Lawrence Livermore National Laboratory

8:00AM C19.00001 High-order energy-stable boundary treatment for finite-difference cut-cell method
NEK SHARAN, PETER BRADY, DANIEL LIVESCU, Los Alamos National Lab — Cut-cell methods simplify grid generation for fluid-flow simulations over complex geometries. Construction of high-order boundary implementation for cut-cell discretization that also provably satisfies stability and conservation constraints, however, remains a challenge, especially for hyperbolic equations. Existing energy-stability proofs of finite-difference methods for initial-boundary value problems require imposing the boundary conditions weakly or by a projection approach, where the computed boundary values may not be exact. Inexact boundary values may be adequate for estimates in certain applications, but they can adversely influence turbulence/mixing statistics in a direct numerical simulation. A framework to prove energy-stability with strong boundary treatment is developed and used to obtain boundary implementation for a Cartesian cut-cell discretization. Linear and non-linear numerical tests to verify the accuracy, stability and conservation properties of the developed method will be discussed.

8:13AM C19.00002 ABSTRACT WITHDRAWN

8:26AM C19.00003 Multidimensional optimization of non-linear shock capturing schemes
RAYNOLD TAN YIYUN, ANDREW OOI, RICHARD SANDBERG, University of Melbourne — In this work, a quasi-linear semi-discrete analysis of shock capturing schemes in multi-dimensional wavenumber space is proposed. Using the dispersion relation of the two dimensional convection and linearized Euler equations, the spectral properties of a spatial scheme can be quantified in two dimensional wavenumber space. A hybrid scheme which combines the merits of the Minimum Dispersion and Controllable Dissipation (MDCD) scheme with the Targeted Essentially Non-Oscillatory (TENO) scheme was developed. Using the proposed analysis framework, the hybrid scheme was spectrally optimized in multidimensional wavenumber space such that the linear part of the scheme can be separately optimized for its dispersion and dissipation properties. In order to demonstrate the improved spectral properties of the new scheme, a series of one dimensional and multidimensional numerical tests were conducted.
8:52AM C19.00005 High-order simulation of flow around geometries on octree meshes using Brinkmann penalization. SABINE ROLLER, HARALD KLIMACH, NIKHIL ANAND, NEDA EBRAHIMI POUR, University of Siegen — Simulation of flow and acoustics over large domains and long distances require highly efficient CFD methods as well as highly scalable implementations on modern supercomputers. High-order Discontinuous Galerkin (DG) implemented on octree meshes fulfill these requirements. Unfortunately, the representation of obstacles in the domain — which usually cause the generation of acoustics — is an issue for high-order methods. In combination with the high order DG (orders up to 32 or 64, i.e. ≈2) the representation of the obstacle surface needs to hold an approximation order equivalent to the scheme order, otherwise the scheme loses its best property at exactly that point. This contribution will investigate the implementation of geometry representations with high-order on a given Cartesian mesh by using a Brinkmann penalization strategy.

9:05AM C19.00006 A numerical strategy for efficient multi-scale and multi-physics simulations using partitioned coupling. NEDA EBRAHIMI POUR, SABINE ROLLER, University of Siegen, Germany — The simulation of complex problems such as multi-scale and multi-physics is still challenging when considering the computational efficiency of those simulations. Simulating all these effects in a single domain is not feasible monolithically, since different scales appear in different areas of the domain, which have to be resolved properly. In order to simulate these kinds of problems in a more efficient way, we make use of partitioned coupling, where we split the domain into subdomains where each of them uses the appropriate set of equations, scheme order and mesh resolution. The subdomains are weakly connected to each other at the boundaries. For the communication and data-exchange between them a coupling approach integrated in our simulation framework APES is used. We will present first results of the coupled scenario and show how well we can reduce the computational cost, when compared to the monolith simulation by means of a small test case, which is still feasible monolithically.

9:18AM C19.00007 A second-order method for convection-diffusion equation across interfaces separated by boundaries of flow. XIANGLONG WANG, University of Michigan and Tulane University, MARK MEYERHOFF, University of Michigan, JOSEPH BULL, Tulane University — Biological mass transport often involves transport across interfaces separated by presence of flow. An example is the recent development of nitric oxide releasing catheters that release nitric oxide into the bloodstream to prevent biofilm formation. The presence of flow often creates large gradients in mass concentration with sharp contrast in Peclet numbers across the interface. Solving such problems with computational methods are challenging, since proper shock capturing methods are essential to resolve these shocks. Our goal is to accurately resolve the shocks present in convection-diffusion problems separated by boundaries of flow. To achieve this goal, we developed a 3D Cartesian-coordinates based method on a model problem simulating release of substance-doped catheters into the bloodstream on a non-orthogonal hexagonal grid. We applied proper directional slope limiting for calculating convection flux and multi-point flux approximation (MPFA) L-method for calculating diffusion flux. This method allows us to achieve stable solutions of the convection-diffusion equation in our model problem with near second-order accuracy for local Peclet numbers up to 5.0. The ability of perform such simulation is essential for guiding the development of nitric oxide releasing catheters.

9:31AM C19.00008 Data-driven optimization strategies for staggered-grid Lagrangian methods. JASON ALBRIGHT, MIKHAIL SHASHKOV, NATHAN URBAN, Los Alamos National Laboratory — For applications dealing with shock waves, algorithmic ingredients like artificial viscosity are essential to avoid highly oscillatory solutions. Yet they also introduce additional model parameters that are usually poorly constrained and consequently are often hand-tuned to problem specific applications. In this talk, we produce optimal values for parameters controlling the amount of artificial viscosity and artificial heat flux through a combination of large ensemble sampling and machine learning-based optimization techniques. We illustrate that the optimal parameter set significantly improves the accuracy, efficiency, and flexibility of the underlying scheme. Although, we illustrate this strategy for a particular discretization scheme, this methodology may be generalized to a much wider variety of existing methods.
9:44AM C19.00009 Reconstructing the piecewise-smooth solution of the Poisson equation for Chebyshev-collocation solution with pointwise exponential convergence. SUDIPTA RAY, SANDEEP SAHA, Department of Aerospace Engineering, Indian Institute of Technology — Computation of an exponentially accurate solution of the Poisson equation which is discontinuous across an interface is restricted by the occurrence of the Gibbs phenomenon. Spectral discretization with inaccurate implementation of the jump conditions produces aphysical oscillations in the numerical solution with algebraic convergence. In the present work, a Chebyshev-collocation spectral discretization is implemented to compute the piecewise-smooth solution of the Laplace and the Poisson equation in two dimensions, where the solution domain contains an interface of complex geometrical shape. The solution is expressed as the sum of a smooth function and a modified Heaviside function at the interface. The unit Heaviside step function is weighted by a smooth jump function which allows the conditions at the interface to be imposed exactly. The modified Heaviside function for interface conditions on the solution and the gradient along the normal is expressed with a weak form expansion. In the presence of global information on the jumps in the form of analytical expressions, the method demonstrates pointwise exponential convergence for the problems considered. In addition, the method appears to be insensitive to perturbations to the interface below the local grid spacing.

9:57AM C19.00010 Reconstructing the piecewise-smooth solution of ordinary differential equations for Chebyshev-collocation solution with pointwise exponential convergence. SUDIPTA RAY, SANDEEP SAHA, SUDIPTA RAY, Department of Aerospace Engineering, Indian Institute of Technology — Physical problems with interfacial discontinuity in the solution or material property are characterized by piecewise-smooth solutions. Numerical computation of problems with interfacial discontinuity requires accurate resolution of the interface conditions. For finite-order methods, the problem may be resolved with local corrections near the interface. Application of spectral methods to approximate the piecewise-smooth solution without an accurate implementation of interface conditions, however, results in the Gibbs oscillations and non-convergent numerical solution. In order to overcome the Gibbs phenomenon, the discontinuous solution is expressed as the sum of a smooth function and a modified Heaviside function at the location of the discontinuity. The unit Heaviside step function is modified with a smooth jump function which exactly satisfies the conditions of discontinuity at the interface. A weak form expansion of the jump function that uses interface conditions up to the first derivative for a second-order ordinary differential equation is proposed. Implementation of a Chebyshev-collocation discretization to problems where the discontinuities in the solution are known in analytic form produces numerical solution that converges exponentially in the maximum norm.

Sunday, November 24, 2019 8:00AM - 10:10AM
Session C20 Geophysical Fluid Dynamics Atmosphere

8:00AM C20.00001 Velocity-defect laws, log law and logarithmic friction law in the convective atmospheric boundary layer\(^1\), MENGJIE DING, CHENNING TONG, Clemson University — The mean velocity profile (MVP) in the convective atmospheric boundary layer (CBL) is derived analytically employing the shear-stress budget equations and the mean momentum equations. The multi-point Monin–Obukhov similarity (MMO) recently proposed provides the scaling properties of the one-point statistics in these equations. Our previous studies have shown that the CBL is mathematically a singular perturbation problem. Thus, we obtain the MVP using the method of matched asymptotic expansions. Three scaling layers are identified: the outer layer, the inner–outer layer and the inner-inner layer. Two new velocity–defect laws are discovered: the mixed-layer velocity–defect law and the surface-layer velocity–defect law. The local-free-convection MVP is obtained by asymptotically matching the expansions in the first two layers. The log law is obtained by matching in the last two layers. The von Karman constant is obtained using velocity and length scales, and therefore has a physical interpretation. A new friction law, the convective logarithmic friction law, is obtained. The present work provides an analytical derivation of the MVP hypothesized in the Monin–Obukhov similarity theory, and is part of a comprehensive derivation of the MMO scaling from first principles.

\(^1\)Supported by NSF

8:13AM C20.00002 Annual variability of atmospheric surface layer characteristics and wind/temperature patterns in Qatar, REZA SADR, YUAN LI, Texas AM University, College Station — Surface wind patterns influence the anemochory, pollutants dissipation. The research of the characteristics of surface layer and turbulence exchange processes can contribute to the local economic construction and understanding of regional plant ecological environment. This work reports on the weather variation in the coastal region of northern Qatar peninsula in the Persian Gulf. Wind velocity, direction, humidity and temperature data for the coastal site of Qatar are recorded from August 2015 to September 2016. Sonic anemometer and weather station data was collected at 9 m height tower. Seasonal wind patterns are analyzed. Shamal wind from Northwest is prevailing for all the four seasons and the annual wind speed is 4.67 m/s. Temperature in June, July, August and September are the highest, with the lowest air pressure and the most occurrence of the summer Shamal. Two other public data are used for comparison with the present data. The normalized variance of wind components and temperature are studied within the framework of Monin-Obukhov similarity theory. Heat and momentum fluxes are calculated and compared with other reported values world-wide.

8:26AM C20.00003 Turbulent characteristics analysis of atmospheric surface layer in coastal region of Qatar, YUAN LI, REZA SADR, Texas AM University, College Station — The turbulence characteristics in coastal region of Qatar are analyzed. The micrometeorological data is collected in the coastal site at (26.08N, 51.36E) by three sonic anemometers located on a 9 m height tower from September 2015 to August 2016. The friction velocity, Obukhov length and the normalized variance of wind components and temperature are studied within the framework of Monin-Obukhov similarity theory. During the 1-year measurement period, 33% of data are characterized as stable. The normalized variances are in agreement to empirical fits from other reported values under both unstable and stable atmospheric stratification. However, the normalized variance of stream wise and transverse velocity at near-neutral condition is slightly higher than other observations while the normalized variance of vertical velocity is slightly lower. The measured wind components and temperature in the surface layer shows the sea breeze circulation. The diurnal pattern characteristics in the coastal region is also analyzed for the onshore and offshore breeze.
Similar Across Thermal Regimes

SCOTT SALESKY, University of Oklahoma, WILLIAM ANDERSON, University of Texas at Dallas — Studies of high-Re wall turbulence have revealed the existence of large scale motions (LSMs) that populate the logarithmic region. These motions are often decomposed into balanced and unbalanced components that represent low-frequency and high-frequency waves, respectively. Such decompositions can be defined, for instance, in terms of eigenmodes of a linear operator. Traditionally these decompositions do not account for phase changes of water since these structures become shorter in the streamwise direction and steepen with increasing buoyancy forcing. Using a suite of large eddy simulations of wall-bounded turbulent shear flows with increasingly unstable thermal stratification, we demonstrate how a balanced–unbalanced decomposition can be performed in the presence of phase changes, by including a slow thermodynamic variable involving total water in addition to a slow potential vorticity variable, which are both associated with the nullspace of the linear operator. An IBL, however, is a non-equilibrium phenomena and is yet to be observed to include the fractal structure.

Forcing and shear instability can give rise to quasi-regular vacillation cycles. The framework is applied to two very distinct geophysical systems, the terrestrial tropical cyclone and the martian polar vortex, where diabatic heating is due to the condensation of water vapor or carbon dioxide, respectively, and where eddy transience gives rise to such effects as eye-wall replacement cycles and controls the polar transport of dust and ice aerosols.

8:52AM C20.00005 Characterizing Intermittency in the Stable Arctic Atmospheric Boundary Layer

MOHAMMAD ALLOUCHE, ELIE BOU-ZEID, Princeton University, JOSE FUENTES, Penn State, MARCELO CHAMECKI, UCLA, OTAVIO ACEVEDO, UFSM, SHAM THANEKAR, Penn State, CEDRICK ANSORGE, U of Cologne — To elucidate the physics of surface-atmosphere exchange processes in Polar Regions, our understanding of the stable atmospheric boundary layer (SABL) where buoyancy damps turbulent kinetic energy needs to advance significantly. We seek to understand the intermittent turbulence regime observed in the strongly stable case. The inertial sublayer, referred to as the atmospheric surface layer (ASL), under such regime is characterized by abrupt transitions between turbulent and laminar states. In this study, we analyze field experimental data from Barrow, Alaska to detect intermittent periods based on non-dimensional statistical metrics. We reveal three clusters of turbulence regimes, two of which correspond to the weakly turbulent periods that feature intermittent behavior (cluster 1: intermittent, cluster 2: transitional) and the third cluster is a fully turbulent regime (cluster 3) only mildly damped by stability. We then investigate the origins of the intermittent bursts based on analyses of the Turbulent Kinetic Energy (TKE) budget equation over these bursts in the TKE time series, and assess the combination of velocity and length scales needed in the eddy diffusion theory under intermittent conditions.

1NOAA-CICS

9:05AM C20.00006 Diabatic vortices: a simple framework for tropical cyclones and the martian polar vortex

RICHARD SCOTT, University of St Andrews — The formation and subsequent evolution of annular distributions of potential vorticity, forced by the combination of diabatic heating and the angular momentum transport of secondary, transverse circulations, is investigated in the framework of the forced shallow water equations at various levels of complexity. Annular potential vorticity is found to develop under a range of forcing conditions depending on the radial location of diabatic heating and the structure of secondary circulation; the simplest, axisymmetric formulation of the model allows the effect of the two forcings to be examined in isolation. Further eddy permitting calculations allow the nonlinear transient evolution to be studied, where the competition of forcing and shear instability can give rise to quasi-regular vacillation cycles. The framework is applied to two very distinct geophysical systems, the terrestrial tropical cyclone and the martian winter polar vortex, where diabatic heating is due to the condensation of water vapor or carbon dioxide, respectively, and where eddy transience gives rise to such effects as eye-wall replacement cycles and controls the polar transport of dust and ice aerosols.

9:18AM C20.00007 Balanced and Unbalanced Components of Atmospheric Flows with Phase Changes of Water

LESLIE SMITH, UW-Madison — Atmospheric variables (temperature, velocity, etc.) are often decomposed into balanced and unbalanced components that represent low-frequency and high-frequency waves, respectively. Such decompositions can be defined, for instance, in terms of eigenmodes of a linear operator. Traditionally these decompositions do not account for phase changes of water since the latter create a piecewise-linear operator that differs in different phases (cloudy versus non-cloudy). Here we demonstrate how a balanced–unbalanced decomposition can be performed in the presence of phase changes, by including a slow thermodynamic variable involving total water in addition to a slow potential vorticity variable, which are both associated with the nullspace of the linear operator with phase changes. Evolution of the fast and slow components of water is illustrated in simulations of moist Boussinesq dynamics, as well as in a turbulent steady-state of the idealized Weather Research Forecasting (WRF) model.

1NSF AGS 1443325

9:31AM C20.00008 Morphological Properties of Large-Scale Motions Remain Self-Similar Across Thermal Regimes

SCOTT SALESKY, University of Oklahoma, WILLIAM ANDERSON, University of Texas at Dallas — Studies of high-Re wall turbulence have revealed the existence of large scale motions (LSMs) that populate the logarithmic layer and modulate the amplitude of small-scale turbulent fluctuations near the wall. In flows with unstable thermal stratification, conventional wisdom states that these structures become shorter in the streamwise direction and steepen with increasing buoyancy forcing. Using a suite of large eddy simulations of wall-bounded turbulent shear flows with increasingly unstable thermal stratification, we demonstrate from instantaneous flow visualizations and conditional averages that morphological properties of these structures remain self-similar with increasing unstable thermal stratification. As thermal stratification increases, the downstream edge of an LSM begins to detach from the wall, leaving a wedge of cool, high-momentum fluid beneath. A simple model is developed for the inclination angle of LSMs with increasing thermal instability; atmospheric surface layer observations from the Advection Horizontal Array Turbulence Study (AHATS) are found to be in good agreement with the predicted inclination angles.
is jointly controlled by the droplet-pool viscosity ratio and the droplet Weber number. Speed of the surface-climbing jet is high, i.e., an order one impact event. Besides the classical Worthington jet, there is another jet before the Worthington jet: the liquid in the pool climbs upward during the impact of a viscous droplet on a less viscous pool in a certain range of the Weber number, i.e., two jets appear successively during pool and tune the impact parameters in a wide range to explore the jet phenomena during droplet impact. We found a two-jet phenomenon when a droplet impacts onto a liquid pool. In this experimental study, we increased the viscosity ratio between the droplets and the liquid liquid pools is a ubiquitous phenomenon in nature and many industrial applications. It is widely known that a Worthington jet may be produced with large viscosity ratios, ZHIZHAO CHE, QUAN DING, TIANYOU WANG, Tianjin University — The impact of droplets on

9:57AM C20.00010 Unmanned Aircraft for Mapping Atmospheric Boundary Layer Induced Geomorphological Changes, VICTORIA NATALIE, JAMEY JACOB, Oklahoma State University — The process of acquiring lower atmospheric measurements onboard unmanned aircraft systems (UAS) is becoming a widely available solution. This study ties atmospheric observations using UAS with photogrammetrically mapped rapid geomorphology to observe and relate the terrain induced effects on lower atmospheric phenomena. Lower atmospheric boundary conditions are characterized through wind velocity and turbulence measurements. In order to achieve this data fusion, empirical data was obtained by UAS mounted anemometers and photogrammetric terrain modeling was acquired utilizing airborne imagery. The results of both are analyzed and related to compare the coupled effects. Wind induced terrain variations are also compared through multiple terrain models, showing the temporal evolution of the landscape. This study was performed at Little Sahara, a State park in Oklahoma, a naturally occurring collection of barchan structures and sand dunes that are heavily influenced by the prevailing southern wind. The datasets are months apart, and the changes in the topography demonstrate how closely linked the atmospheric movement and the changes in topography are to each other. Wind observations were made with multiple direct and indirect methods and are presented.

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C21 Drops: Impacts with Liquids I 603 - Jim Duncan, University of Maryland

8:00AM C21.00001 Two-jet phenomenon during the droplet impact on liquid pools with large viscosity ratios, ZHIHAO CHE, QUAN DING, TIANYOU WANG, Tianjin University — The impact of droplets on liquid pools is a ubiquitous phenomenon in nature and many industrial applications. It is widely known that a Worthington jet may be produced when a droplet impacts onto a liquid pool. In this experimental study, we increased the viscosity ratio between the droplets and the liquid pool and tune the impact parameters in a wide range to explore the jet phenomena during droplet impact. We found a two-jet phenomenon during the impact of a viscous droplet on a less viscous pool in a certain range of the Weber number, i.e., two jets appear successively during one impact event. Besides the classical Worthington jet, there is another jet before the Worthington jet: the liquid in the pool climbs upward along the surface of the droplet, and finally collides on the apex of the droplet, resulting in a ‘surface-climbing’ jet. The two-jet phenomenon is jointly controlled by the droplet-pool viscosity ratio and the droplet Weber number. Speed of the surface-climbing jet is high, i.e., an order of magnitude higher than the impact speed.

8:13AM C21.00002 Effects of wind on the generation of secondary droplets and ring waves due to drop impact onto a water surface1, XIANYI LIU, YANG LIU, JAMES H DUNCAN, Department of Mechanical Engineering, University of Maryland, College Park, AIR-SEA INTERACTION TEAM — The effects of wind on the generation of secondary droplets and ring waves during the impact of a single water drop on a deep-water surface are studied experimentally in a wind tunnel that has a test section with a water pool. Experiments are performed by varying impacting drop diameters ranging from 2.0 to 4.0 mm and wind speeds up to 10 m/s. Secondary droplets and ring waves generated during drop impact are measured with a backlit, cinematic shadowgraph technique that employs a high-speed camera. Our experimental results show that after the drop hits the water surface an asymmetrical crown forms on the leeward of the droplet while a wave swell forms on the windward. Secondary droplets are generated from the crown rim. It is found that the diameters and velocities of these secondary droplets are drastically changed with the wind speed. The capillary ring waves on the windward side of the drop impact are stronger than those on the leeward side.

1This work is supported by the Division of Ocean Science, National Science Foundation, under award number 1829943.

8:26AM C21.00003 Understanding thin air film entrapment in drop-pool impact events via numerical simulations1, SHAHAB MIRJALILI, ALI MANI, Stanford University — Experiments have shown that when an O(1mm) water drop impacts a water pool with O(1m/s) velocity, hundreds of micro-bubbles can be entrained in a process known as Mesler entrainment. These bubbles are remnants of a thin extended air film that is entrapped between the two liquid bodies prior to their contact. Despite such observations, neither the details of the mechanism of Mesler entrainment, nor the requirements for it to happen have been established. In this work, we numerically study the impact of a drop on a deep pool to understand the evolution of the liquid surfaces and air film. Our simulations, alongside analytical arguments, lead to the discovery of a capillary transition in the dynamics of the thin air film that allows for formation of high aspect ratio films that can shed micro-bubbles. Based on this observation, we claim that the occurrence of this transition is a trade-off between capillary forces and Van der Waals forces, leading to a criterion for delineating the boundaries of the Mesler entrainment regime in the parameter space. By discovery of scaling laws governing the film thickness after transition, we pave the path for quantitative prediction of the micro-bubbles that are shed as the film retracts after rupture.

1Supported by ONR
8:39AM C21.00004 Impact of immiscible drop onto a pool: jetting and sound generation

ZIQIANG YANG, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology — We use high-speed video imaging to study the impact of a perfluorohexane drop on a water pool, with focus on bubble entrainment and fine vertical jetting during the crater collapse. The drop stretches out at the bottom of the crater, before collapsing during its rebound, often entrapping a small bubble inside the drop liquid, or shooting out a fine vertical jet. The entrapment regimes are mapped in the Weber-Froude number space. The size of entrapped bubble scales with impact Weber number. We observe a sequence of critical conditions for the formation of fine jets, which emerge at velocities up to 32 m/s and jet thickness of a few microns, which break up into as many as 50 micro-droplets. The fastest jets, at 45 m/s, occur for a novel multi-dimple crater shape without bubble pinch-off. We also investigate the sound production from the jetting during the crater collapse. The drop stretches out at the bottom of the crater, before collapsing during its rebound, often entrapping a small bubble inside the drop liquid, or shooting out a fine vertical jet. The entrapment regimes are mapped in the Weber-Froude number space. The size of entrapped bubble scales with impact Weber number. We observe a sequence of critical conditions for the formation of fine jets, which emerge at velocities up to 32 m/s and jet thickness of a few microns, which break up into as many as 50 micro-droplets. The fastest jets, at 45 m/s, occur for a novel multi-dimple crater shape without bubble pinch-off. We also investigate the sound production from the bubble oscillations following their pinch-off. By synchronizing hydrophones with the high-speed imaging, we connect the phase of the bubble oscillation with the acoustic signal. We find good agreement with the Minnaert frequency when using the fluid properties of the drop. This applies reasonably well even for very small bubbles where the frequency approaches 100 kHz.

1 This work was funded by King Abdullah University of Science and Technology (KAUST) under Grant No. URF/1/2621-01-01.

8:52AM C21.00005 Singular jets from the collapse of craters at a pool surface

YUANSHI TIAN, SIGURDUR THORODDSEN, KAUST — The collapse of drop-impact craters can generate a fast singular jet from a dimple which forms at its bottom. A finite-time singularity in the bottom curvature of the crater has in the past been considered as the cause of this singular jet. Self-similar capillary-inertial solutions predict that the radius of the cavity will collapse with time, as \( t \rightarrow 0 \) to the power \( 2/3 \). However, Thoroddsen et al.[1] used a high-speed camera to demonstrate that the final collapse has a power-law closer to a purely inertial collapse, with \( R \sim t^{1/2} \). They also observed no curvature singularity. Herein, we use two synchronized high-speed cameras to study the dimple collapse, at even higher time resolution. One is an ultra-fast camera, capable of up to 5 Mfps, which tracks the crater collapse. The second high-speed camera captures the corresponding speed of the singular jet, at up to 400 kfps. The experiment is performed inside a vacuum chamber to control the ambient pressure. The fastest velocity of singular jets is found to be around 130 m/s and occurs at reduced pressure without crater pinch-off.


KAUST Grant No. URF/1/2621-01-01

9:05AM C21.00006 Study on a vertically falling droplet toward a liquid pool with an inclined bottom wall

YEAWAN LEE, Department of Mechanical Engineering, KAIST, YOUNGDO KIM, Mueunjae School of Undergraduate Studies, POSTECH, HYOUNGSOO KIM, Department of Mechanical Engineering, KAIST — We investigate the evolution of liquid interface deformation after a vertically falling droplet impacts to a liquid pool with an inclined bottom wall. We observed that initially an almost hemispherical cavity was formed, and then asymmetric cavity reversal was observed. Eventually, a tilting jet was measured. For the systematic experiments, substrate angle, depth, droplet diameter, impact velocity, surface tension, and viscosity were varied. We found out that the hemispheric cavity development, which is driven by inertia, is analogous to the impact problems in a deep bath. Next, when the cavity retracts, it shows an oblique conical shape, which is due to the different wave propagation mechanism depending on the bath depth, i.e. a relatively shallow and deep bath. This purely geometry-induced effect causes the tilting jet. Finally, we provided a theoretic model to predict the jet inclination angle by assuming that two different wave propagation competed. Furthermore, we controlled the jet direction by changing the boundary condition of the inclined substrate.

1 This work was supported by Basic Science Research Program through the National Research Foundation of Korea funded by the Ministry of Science (NRF-2018R1C1B6004190) and BK21 Plus program. We also thank that this paper is based on research which has been conducted as part of the KAIST-funded Global Singularity Research Program for 2019.


FAN-WEI WANG, CHUN-TI CHANG, National Taiwan University — What happens when a drop impacts a pool with a depth comparable to the drop’s diameter? Through high-speed imaging, our experiments reveal three categories of patterns: fingers at lower impact velocities, flowers at higher impact velocities, and stagnant rings in between. Recent observations will be reported in this talk, together with our characterization by scaling analysis.

Ministry of Science and Technology, Taiwan, Grant: No. MOST 107-2218-E-002-070-MY3

9:31AM C21.00008 Direct numerical simulation of rain drop impact on a thin layer of oil over a deep water pool

FRANCIS OGOKE, WOUTER MOSTERT, Princeton University, MARIE-JEAN THORAVAL, Xi’an Jiaotong University, LUC DEIKE, Princeton University — The impact of a water droplet onto a deep pool coated by a film of oil has not yet been thoroughly investigated numerically in the large Weber number range. This process occurs during rainfall on oil slicks at sea, and ejects oily aerosols into the atmosphere that later forms atmospheric particulates. We present direct numerical simulations of the three-phase process using the solver Basilisk. The numerical results are qualitatively and quantitatively compared to existing experimental data, and discuss the influence of numerical resolution on the crown and canopy closure. Finally, the effects of the oil properties and drop shape upon impact on the resulting splash dynamics are investigated.

9:44AM C21.00009 The impact of an oil droplet on oil layers on water bath

DOHYUNG KIM, Sungkyunkwan University, ILDOO KIM, FG Research LLC, JINKEE LEE, Sungkyunkwan University — The impingement of droplets onto another liquid has been investigated under various settings depending on applications. In this work, we investigate the impact of an oil droplet on oil layers on water to simulate the action of the dispersant applied from ship or aircraft to remove the oil contamination on water. Our experiments cover a range of Weber number from 50 to 1000, and we observe the morphological change of the fluid interfaces using the high-speed video imaging. From the image analysis, the length scales of impact craters are measured with respect to the droplet size, the impact velocity, and the thickness of oil layer. Our measurement indicates that the impact dynamics depends on the thickness of oil layer in a non-monotonic manner, which we rationalize using a hypothesized model.

JL thanks to the support by KEITI (2019002790003).
of azimuthal instabilities. and the downward liquid jet that pierces through the bottom of the cavity, are analyzed and discussed, with a specific focus on the inception involved, such as the crown formation above the cavity created by the impact, the ligaments emanating from the rim at the top of the crown, and the downward liquid jet that pieces through the bottom of the cavity, are analyzed and discussed, with a specific focus on the inception of azimuthal instabilities.

**Sunday, November 24, 2019 8:00AM - 10:10AM –**

**Session C22 Drops: Interaction with Elastic Surfaces, Particles, and Fibers**  
**604 - Farzad Ahmadi, Virginia Tech**

**8:00AM C22.00001 Elastocapillary flow driven blood-plasma separation.**  
**ALWAR SAMY RAMASAMY, ASHIS KUMAR SEN, Indian Institute of Technology (IIT) - Madras — We report a lab-on-a-membrane device which can separate plasma from whole blood due to a combined effect of capillary flow and sedimentation of blood cells. The initial length of the polydimethylsiloxane (PDMS) membrane is made hydrophilic by exposing to oxygen plasma and it is stuck to the PDMS substrate forming a 90° edge. As the dispensed whole blood comes in contact with the membrane, force due to surface tension deflects the membrane and the consequent evolution of blood meniscus results in a Laplace pressure drop that drives the flow. The high velocity gained in the hydrophilic region is sufficient to drive the flow in to the hydrophobic region. As the flow advances, sedimentation of blood cells occurs along the length resulting in cell free layer of plasma. The geometry of the membrane and the hydrophilic length are adjusted such that the time scale required for sedimentation of blood cells is smaller than the flow time scale. As a proof of concept, a detection strip is embedded with the device to detect the level of glucose present in blood plasma.

**8:13AM C22.00002 The droplet on a sugar fiber**

**STEPHANE DORBOLO, FNRS-University of Liege, FLORIANE WEYER, NICOLAS VANDEWALLE, University of Liege, ALEXANDRE DELORY, ESPCI, ESPCI TEAM, FNRS-UNIVERSITE DE LIEGE TEAM — The shape and the motion of a droplet on a fiber are determined by the volume of the droplet, the radius of the fiber and the contact angle (static, advancing and receding) of the liquid on the material of the fiber. We consider a particular case in which the radius of the fiber is modified by the very presence of the droplet. The chosen system consists of a fiber made of glucose on which a water droplet is released. When the fiber is vertical, the droplet slides down the fiber before stopping at the extremity of the fiber. At this point, the droplet dissolves the fiber until the droplet moves upwards. This surprising motion of the droplet is analyzed regarding the dissolution dynamics.**

**8:26AM C22.00003 Redwood-inspired fog harps**

**WEIWEI SHI, THOMAS VAN DER SLOOT, BRANDON HART, BROOK KENNEDY, JONATHAN BOREYKO, Virginia Tech — In California, coastal redwoods (*Sequoia sempervirens*) obtain 34% of their annual hydrologic input from fog drip, as fog droplets are able to effectively slide along the parallel needle arrays to fall onto the soil. Inspired by the redwoods, we recently developed anti-clogging “fog harps” comprised of an array of vertically oriented wires that harvested 2-3X more water compared to traditional meshes during experiments with scale-model in lab. Here, we conduct outdoor field tests for a full-scale fog harp placed side-by-side with a mesh harvester. The harp harvested anywhere from 5X to 70X more water compared to the mesh, depending upon the weather conditions. This enhancement is attributed to the harp’s minimal contact angle hysteresis along the drainage pathway, which completely prevents clogging and even allows for water collection in subpar fog conditions. On the harp, droplets tended to slide along a single wire at a critical volume of only about 0.1 mm³. In contrast, most of the fog collected by the mesh remained pinned, resulting in its gradual evaporation. We expect that the fog harp’s unprecedented collection efficiency will expand the regions where fog harvesting is a viable means of water harvesting.**

**8:39AM C22.00004 Impact and imbibition of blood drops with textiles: Where does this stain come from?**

**DANIEL ATTINGER, RICHARD FAFLAK, Iowa State University — Forensic investigators are sometimes asked if a given stain on fabric could have originated from a blood source at a specific relative location. A wide range of values of the maximum distance that a blood drop can travel have been reported, from less than one meter to more than 10 meters. The formation of bloodstains on fabrics can involve fast atomization mechanisms, flights of deforming droplets, and capillary interactions of a complex fluid with a multiscale substrate. Here we solve the above forensic question with fluid dynamics and inverse data search. Fluid dynamic simulations involve Newton’s equation of motion, models for aerodynamic drag forces on deformable particles, and an in-house capillary model of blood wicking in fabrics. Key parameters are the drop size, launch velocity and launch angle. It is assumed that the ambient air is quiescent, and that stains exhibit similar areas on the front and back of the fabric. The latter assumption is verified for stains larger than the fabric thickness. Simulation results are then mined for the stain size, a parameter directly measurable on a crime scene. Experiments of blood spattered on fabric are in agreement with the simulation results. Findings are presented in a simple chart which discriminates whether a stain of specific size, on a specific fabric, can originate from a blood source at a specific relative location, or not.

**Support is acknowledged from the Army Research Office and Defense Forensic Science Center, award W911NF1520111.**
A model of droplet durotaxis driven by the elastocapillary response of a soft viscoelastic substrate
dynamic wetting of soft solids has many industrial and biological applications which require a rigorous understanding of the underlying fluid transport mechanism. One such phenomena, known as durotaxis, involves the spontaneous motion of liquid drops on a soft substrate with a thickness gradient. A passive driving force is generated by the elastocapillary deformation of the compliant substrate due to the interaction with the drop. We model the interactions of a two-dimensional drop with a neutrally-wetting viscoelastic substrate with thickness gradient. The substrate response is characterized by a sharp triangular wetting ridge at the contact line whose geometry changes the macroscopic contact angle of the liquid drop from its equilibrium value. The gradient in substrate thickness causes the contact angle to be less on the thicker side, which generates a driving force that moves the drop in the direction of increasing thickness. The drop velocity is determined by the viscoelastic relaxation of the substrate (viscoelastic braking), and computed from a self-consistent model that relates the soft wetting force to the viscoelastic dissipation. We find our results to be in close agreement to the velocities observed in experiment by Style et al. 2013, PNAS.

1 Funding support from NSF Grant CBET-1750208 is acknowledged.
8:00AM C24.00001 Effect of topology changes on the breakup of a periodic liquid jet, ALBERTO ROMAN-APANADOR, University Of Notre Dame, STEPHANE ZALESKI, Sorbonne Universite, GRETA T. TRYGGVASON, JIACAI LU, Johns Hopkins University — The breakup of a periodic jet is examined computationally, using a front-tracking/finite-volume method, where the interface is represented by connected marker points moving with the fluid, while the governing equations are solved on a fixed grid. Tracking the interface allows control of whether topology changes take place or not. The Reynolds and Capillary numbers are kept relatively low so most of the flow is well resolved. The effect of topology changes is examined by following the jet until it has mostly disintegrated, for different “coalescence” criteria, based on the thickness of thin films and threads. It is found that while the overall breakup is relatively insensitive to the exact conditions for topology changes, various quantitative measures of the evolution are not. The evolution of both two-dimensional and fully three-dimensional flows is examined. It is found that although there is a significant difference between the evolution when no breakup takes place and when it does, once breakup takes place the evolution is relatively insensitive to exactly how it is triggered for a range of coalescence criteria. However, excessively aggressive coalescence leads to different outcomes.

8:13AM C24.00002 A robust multiphase flow solver for high density ratio wave structure interaction problems, AMNEET PAL S. BHALLA, San Diego State University — In this talk, we present a robust, adaptive numerical scheme for simulating high-density ratio and high shear multiphase flows on locally refined staggered Cartesian grids that adapt to the evolving interfaces and track regions of high vorticity. The algorithm combines the interface capturing level set method with a variable-coefficient incompressible Navier-Stokes solver that is demonstrated to stably resolve material contrast ratios of up to six orders of magnitude. The discretization approach ensures second-order pointwise accuracy for both velocity and pressure with several physical boundary treatments, including velocity and traction boundary conditions. To ensure the stability of the numerical scheme in the presence of high density and viscosity ratios, we employ a consistent treatment of mass and momentum transport in the conservative form of discrete equations. We demonstrate through several test cases that the lack of consistent mass and momentum transport in non-conservative formulations, which are commonly used in practice can yield very large numerical error and very poor accuracy for convection-dominant high-density flow regimes. Finally, we combine our robust multiphase fluid solver with an efficient implementation of immersed boundary method to simulate wave-structure interaction problems. Several cases from practical ocean and marine engineering applications, including simulations of wave energy converter devices are presented.

8:26AM C24.00003 Efficient, Discretely Conservative Multi-Material Phase Transport on Unstructured Meshes Using the Interface Reconstruction Library, ROBERT CHIODI, Cornell University, PETER BRADY, NEIL CARLSON, Los Alamos National Laboratory — The Truchas Multiphysics solver was developed at Los Alamos National Laboratory in order to simulate advanced metal casting processes of exotic materials. These processes involve complicated mold geometries that require the use of unstructured meshes in order to adequately resolve the shape of the mold. To enable the accurate simulation of the filling of these molds with liquid metal, we have implemented an unsplit geometric multi-material volume of fluid method capable of individually tracking the many constituent ingredients involved in metal casting. This was done using a newly developed, open-source library, the Interface Reconstruction Library, which handles the difficult computational geometry and interface reconstruction methods needed in geometric volume of fluid methods. In this talk, we will detail how to implement an efficient algorithm for discretely conservative multi-material phase advection on unstructured meshes. Its computational performance and solution accuracy on both hexahedron and tetrahedron meshes will be compared to the current state-of-the-art for a suite of canonical test cases. This abstract is approved for release under LA-UR-19-27091.

8:39AM C24.00004 Two-way coupled Euler-Lagrange simulations with particles and mesh spacing of arbitrary sizes, BEREND VAN WACHEM, FABIAN DENNER, FABIEN EVRARD, Otto-von-Guericke University Magdeburg — The Euler-Lagrange (EL) approach is widely used to simulate particulate flows, because of the relatively low computational cost and the straightforward modelling of particle-particle interactions. Since relying on an assumption of separation of scales between the general features of the flow (resolved on the underlying fluid mesh) and those at the scale of the particles (not resolved on the mesh), the EL method typically requires tracked particles to be much smaller than the grid cells in which the flow equations are solved, commonly referred to as the “particle-in-cell” approach. For the cases that require high fluid mesh refinement, this can strongly limit the size of the particles that can be accurately tracked. In this work, we propose an EL approach that alleviates the particle size restrictions. It relies upon the filtering of the flow equations with a particle marker function; a process in which a length-scale is chosen. We also present a model for the reconstruction of the undisturbed flow velocity and volume fraction at the particle locations, based on the study of the flow through a regularised momentum source and/or a volume fraction dimple. The ability of the proposed framework to accurately track particles of arbitrary sizes is shown with an array of test-cases.

8:52AM C24.00005 A hybrid PIC-DEM approach for multi-phase computational fluid dynamics, ROBERTO PORCU, ANN ALMGREN, MICHELE ROSSO, Lawrence Berkeley National Laboratory, JORDAN MUSSER, National Energy Technology Laboratory, WILLIAM FULLMER, National Energy Technology Laboratory; Leidos Research Support Team, ANDREW MYERS, OSCAR ANTEPARA, Lawrence Berkeley National Laboratory — MFiX-Exa is a new code being developed by the National Energy Technology Laboratory and Lawrence Berkeley National Laboratory as part of the U.S. Department of Energy’s Exascale Computing Project. MFiX-Exa originated by combining the discrete element method (DEM) modules of the classic MFiX code (mfix.netl.doe.gov) with a modern low Mach number projection method for the continuous fluid phase. The new algorithm is implemented using the AMReX software framework for massively parallel block-structured applications (amrex-codes.github.io). Despite the ever-increasing computational power offered by world-leading supercomputers, DEM is still prohibitively expensive for the modeling of large industrial-scale problems. Other methods, as particle-in-cell (PIC), are less computationally intensive, but they tend to be less accurate than DEM. In this work, we exploit the efficiency of PIC and accuracy of DEM to introduce a hybrid multi-phase PIC-DEM approach. Overall, the strategy relies on applying the PIC model to particle-dense regions while keeping the DEM model for the dilute parts of the domain. Within this setup, the modeling of the PIC/DEM transitions is proposed. The method and results are to be presented in the context of chemical loop reactors simulations.

1Los Alamos National Laboratory is operated by Triad National Security, LLC, for the National Nuclear Security Administration of the U.S. Department of Energy (Contract No. 89233218CNA000001)

1This research was supported by the Exascale Computing Project (17-SC-20-SC), a collaborative effort of the U.S. Department of Energy Office of Science and the National Nuclear Security Administration.
Simulation, YUNCHAO YANG, S. BALACHANDAR, University of Florida — We developed a highly scalable immersed boundary
method (IBM) algorithm for multiphase flow simulation. In particular, a Double Bin Ghost Particle (DBGP) algorithm is designed to obtain the
distributed data storage and scalable data transferring features for the particle phase. The proposed algorithm uses a New Queen/Worker data
structure to indicate the particle-level and marker-level quantities and communications. In the DBGP algorithm, each particle is represented
by a queen marker and surface worker markers. The queen marker determines the motion and total force of a particle. The worker marker
performs the fluid-particle interaction, including velocity interpolation and force projection. A double binning system is determined through the
Cartesian binning of the physical domain to relate each MPI rank with its overlapping bins. By searching the bin-to-rank map, all remote MPI
ranks influenced by the local particle can be readily identified. The ghost queen marker is generated in the remote MPI rank based on the local
queen marker and the queen binning structure, and same for the ghost worker maker. Experimental results show that the algorithm is efficient
and scalable on large-scale computations.

Techniques for sharp numerical simulations of incompressible
multiphase flows1. RAPHAEL EGAN, FREDERIC GIBOU, University of California, Santa Barbara — We present recent progress
and developments to enable numerical simulations of incompressible two-phase flows in a fully sharp fashion, avoiding smearing of any fluid
properties across interfaces. We use distributed adaptive cartesian Quadtree/Octree grids with a levelset method to represent the interface(s).
The ability to solve elliptic interface problems with sharp jump conditions and discontinuous coefficients is essential to such applications. We
discuss the corresponding numerical challenges and illustrate them with respect to the governing jump conditions and the balance of viscous
stress across the interface. We present numerical methods to address these challenges and to ensure accurate solutions in infinity norm (i.e.,
even for points close to the interface) for accurate interface transport.

Numerical simulations of the full ink-jet printing processes: From
jetting to evaporation1. CHRISTIAN DIDDENS, YAXING LI, TIM SEGERS, University of Twente, HANS REINTEN, YOURI
DE LOORE, HERMAN WIJSHOFF, Oc-Technologies B.V., DETLEF LOHSE, University of Twente — Ink-jet printing requires to perfectly
control both the jetting of droplets and the subsequent droplet evaporation and absorption dynamics. Considerable complexity arises due to
the fact that ink is constituted of a mixture of different liquids, surfactants and pigments. Using a sharp-interface ALE finite element method, we
numerically investigate the main aspects of ink-jet printing, both on the jetting side and on the drying side. We show how a short pause
in jetting can result in clogged nozzles due to solvent evaporation and discuss approaches how to prevent this undesired phenomenon. Once
the droplets have been jetted on paper and is evaporating, the print quality can be deteriorated by the well-known coffee-stain effect, i.e., the
preferential deposition of particles near the rim of the droplet. This can be prevented in several ways, e.g., employing controlled Marangoni flow
via surfactants or co-solvents or printing on a primer layer jetted in beforehand, thus creating a homogeneous deposition pattern for a perfect
final printout.

This work is part of an Industrial Partnership Programme of the Netherlands Organisation for Scientific Research (NWO), which is
co-financed by Oc-Technologies B.V., University of Twente, and Eindhoven University of Technology.
A Numerical Formulation to Study Interactions Between Fluids and Deforming Solid. JIAZHEN QIAO, AMIR RIAZ, University of Maryland, College Park, AKASH DHURU, ELIAS BALARAS, George Washington University — In the present work, level set formulations are used to track solid-fluid interface as well as to track a dynamic grid which captures solid deformation. The solid is assumed to be viscoelastic and a unified framework of equation of motion is used to solve for both solid and fluid dynamics. Fluid-Structure Interaction is accounted for by using an external body force term to enforce no-slip boundary condition at the interface and an elastic stress term to impose elastic stress boundary condition at the interface. The elastic stress term is implemented through a pressure jump which corrects the pressure distribution as well as the velocity field in the computational domain as a result of the presence of the elastic solid. The proposed method has an advantage of robust implementation in three dimensions and has the potential of incorporating different viscoelastic models to account for various material properties.

8:13 AM C25.00002 Advancements to a Dual-Scale approach for Simulating Turbulent Phase Interface Interactions. DOMINIC KEDELTY, MARCUS HERRMANN, THOMAS ZIEGENHEIN, Arizona State University — Direct Numerical Simulation remains an expensive task for atomization simulations. To decrease the burden of DNS, a dual-scale modeling approach (Gorokhovski and Herrmann, 2008) that describes turbulent phase interface dynamics in a Large Eddy Simulation framework to study the evaporation process of a liquid in an inert gas using the volume of fluid method. The proposed methodology successfully addresses the two main challenges in performing direct numerical simulation of phase-changing flows when a whole-domain formulation is adopted: the interface-normal velocity jump and the accurate calculation of the interfacial mass-flux exchanged between the two phases. The former is handled by constructing a continuous and divergence-free liquid velocity field, which is used to compute the interface velocity, while the latter is accomplished by reconstructing a level-set function. This resolved realization is maintained using a Refined Local Surface Grid approach (Herrmann, 2008) employing an unsplit geometric Volume-of-Fluid method (Owkes and Desjardins, 2014). Advection of the phase interface on this DNS scale requires a reconstruction of the fully resolved interface velocity. In this work, results from the dual-scale LES model employing sub-filter turbulent eddy reconstruction by combined approximate deconvolution and non-linear spectral enrichment (Bassenne et al. 2019) and sub-filter surface tension model (Herrmann 2013) are compared to DNS results for a phase interface in a homogeneous isotropic turbulent flow at two different Weber numbers.

The support of NASA TT grant NNX16AB07A is gratefully acknowledged.

8:26 AM C25.00003 A volume of fluid method for interface-resolved simulations of evaporating flows. NICOLÒ SCAPIN, PEDRO COSTA, LUCA BRANDT, Kungliga Tekniska högskolan — We developed a numerical framework to study the evaporation process of a liquid in an inert gas using the volume of fluid method. The proposed methodology successfully addresses the two main challenges in performing direct numerical simulation of phase-changing flows when a whole-domain formulation is adopted: the interface-normal velocity jump and the accurate calculation of the interfacial mass-flux exchanged between the two phases. The former is handled by constructing a continuous and divergence-free liquid velocity field, which is used to compute the interface velocity, while the latter is accomplished by reconstructing a level-set function. This resulting approach is built on top of an efficient, FFT-based two-fluid Navier-Stokes solver coupled with an algebraic volume of fluid method (MTHINC), and extends with the corresponding transport equations for the vaporized liquid mass and thermal energy. The method was thoroughly tested against benchmarks of increasing complexity, which show its excellent mass conservation properties and good overall performance.

8:39 AM C25.00004 Quadric interface reconstruction from volume fractions for curvature estimation. AUSTIN HAN, OLIVIER DESJARDINS, Cornell University — In this talk, we present a new method for evaluating the curvature of a captured interface in the context of volume of fluid (VOF). This method endeavors to fit a paraboloid to the interface using only volume fraction data in a neighborhood of cells. It shares strong similarities with the height-function (HF) method, which also represents the interface as a paraboloid. But because the HF method orients the paraboloid along the directions of the underlying Cartesian mesh, its accuracy deteriorates severely in cases of poorly resolved slanted interfaces. In contrast, we allow here for the paraboloid to be arbitrarily rotated and propose a method for evaluating the volume of each cell capped by the arbitrary paraboloid. We verify the performance of this approach on a range of canonical problems, paying special attention to the dependence of the error on the alignment of the interface with the mesh. Finally, we discuss the extension of this method to unstructured meshes.

On algebraic TVD-VOF methods for tracking material interfaces. SERGIO PIROZZOLI, SIMONE DI GIORGIO, Sapienza University of Rome, Department of Mechanical and Aerospace Engineering, ALESSANDRO IAFRATI, CNR-INSEAN, Istituto Nazionale per Studi ed Esperienze di Architettura Navale — We revisit simple algebraic VOF methods for advection of material interfaces based of the well established TVD paradigm. We show that greatly improved representation of contact discontinuities is obtained through use of a novel CFL-dependent limiter whereby the classical TVD bounds are exceeded. Perfectly crisp numerical interfaces are obtained with very limited numerical atomization (flotsam and jetsam) as compared to previous SLIC schemes. Comparison of the algorithm with accurate geometrical VOF shows larger error at given mesh resolution, but comparable efficiency when the reduced computational cost is accounted for.

This work has been financially supported by the Flagship Project RITMARE The Italian Research for the Sea, coordinated by the Italian National Research Council and funded by the Italian Ministry of Education, University and Research within the National Research Program 20142015.

9:05 AM C25.00006 Height-function method for curvature estimation on two- and three-dimensional non-uniform Cartesian grids. FABIEN EVRARD, FABIEN DENNER, BEREND VAN WACHEM, Otto-von-Guericke University Magdeburg — Estimating the curvature of an interface from the corresponding discrete indicator function field is known as one of the main challenges associated with the simulation of interfacial flows with surface tension. The sharpness of the discrete indicator function renders classical differenciation approaches ineffective (understand: divergent) as the mesh resolution is increased. Out of all the methods available in the literature, the height-function (HF) method is the sole example of a curvature estimation technique that converges with mesh refinement and has been shown to exhibit performances that are superior to other approaches when coupled to a multiphase flow solver. Although a second-order accurate general formulation for non-uniform Cartesian grids has been derived for the two-dimensional case, the three-dimensional HF formulas have so far been limited to uniform Cartesian grid configurations, where they correspond to classical finite difference formulas. This work presents an extension of the well-known height-function curvature estimation method to non-uniform Cartesian grids, both for the two- and three-dimensional cases. The convergence of the method is studied through its application to known analytical surfaces, and ways to improve its order are discussed.
9:18AM C25.00007 A diffused-interface approach for simulating compressible multiphase flows within an adaptive mesh refinement framework, KARTHIK KANNAN, Arizona State University, FABIAN FRITZ, NICO FLEISCHMANN, Technical University of Munich, CARLOS BALLESTEROS, MARCUS HERRMANN, Arizona State University — A diffused-interface method for solving the compressible, multicomponent Navier-Stokes equations, i.e. the quasi-conservative five-equation model (Allaire et al., 2002) including capillary and viscous effects (Coralic and Colonius, 2014, Garrick et al., 2017) is used in conjunction with a novel, unstructured, cell-based adaptive mesh refinement (AMR) library (Ballesteros, 2019). Low dissipation, high spatial resolution is obtained by using a WENO-Z (Borges et al., 2008) discretization, while the numerical smearing of the material interface is controlled using a THINC scheme (Shyue and Xiao, 2014). This ensures higher-order while limiting the material interface width to 2-3 mesh cells. Surface tension effects are incorporated by means of a modified HLLC solver (Garrick et al., 2017). Curvature is computed using a stretched variant of the standard height function method (Cummings et al., 2005) to account for the smearing of the material interface. The use of AMR helps achieve higher mesh resolution in regions of liquid and shock discontinuities, which is crucial to compressible atomization applications. Results for test cases ranging from shock-interface interaction to liquid atomization are presented.

9:31AM C25.00008 Surface tension effects on the evolution of interfaces in multi-material compressible flows, PEDRAM BIGDELOU, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — We report on the effect of surface tension on the evolution of perturbed material interfaces in compressible multimedium flows. The level set method is used to track the interface, while the real Ghost Fluid Method1 (gFGM) captures the interfacial coupling between the fluids. We implement these techniques in an in-house code IMPACT, designed for simulation of compressible flows with shocks and material interfaces. We present various test problems to address how surface tension affects the growth of perturbed interfaces driven by shocks. In particular, we examine surface tension effects on the baroclinically driven Richtmyer-Meshkov instability.2,3 The 2D simulations were initialized with a sinusoidal perturbation imposed at the interface. An incident shock crosses the interface, followed by the growth of the imposed perturbation. The simulations were conducted at different values of the surface tension, and the variation in the instability growth rate was compared with recently proposed models.2 Wang C.W., Liu T.G., Kho P.C., SIAM J. Sci. Comput. 28(1) (2006) 278-302. 3 Richtmyer R.D., Commun. Pure Appl. Math. 13 (1960) 297-319. 3) Meshkov E.E., Fluid Dyn. 4 (1969) 101-104.

9:44AM C25.00009 ABSTRACT WITHDRAWN —

9:57AM C25.00010 Non-local surface tension model for N-phase flows1, AMANDA HOWARD, ALEXANDRE TARTAKOVSKY, Pacific Northwest National Laboratory — We propose a nonlocal model for surface tension obtained in the form of an integral of a molecular-force-like function added to the Navier-Stokes momentum conservation equation for N-phase fluid flows. We demonstrate that our model recovers both microscale and macroscale features of multiphase flow, eliminating the need for expensive hybrid MD-NS models, and providing strong advantages for modeling multiphase flows at length scales not feasible with MD simulations. We present benchmark cases for the nonlocal model with comparisons to the level set method for N-phase flows and fluid-fluid-solid flows. Results are shown to be in agreement with analytical and previous numerical results.

1This work was supported by the U.S. Department of Energy (DOE) Office of Science, Office of Advanced Scientific Computing Research as part of the New Dimension Reduction Methods and Scalable Algorithms for Nonlinear Phenomena project. Pacific Northwest National Laboratory is operated by Battelle for the DOE under Contract DE-AC05-76RL01830.

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C26 Drag Reduction II

8:00AM C26.00001 Drag reduction of three-dimensional riblets on a flat plate turbulent boundary layer1, GIOACCCHINO CAFIERO, GAETANO IUSO, Politecnico di Torino — We performed an experimental investigation of the turbulent flow on a flat plate presenting micro roughness. In addition to the typical longitudinal micro-grooves, commonly referred to as riblets, we also investigate the performance of three-dimensional riblets, i.e. presenting a sinusoidal pattern. Further to the typical cross-section parameters that characterize longitudinal riblets, namely depth (h) and spacing (s) of the micro grooves, the sinusoidal riblets add two more parameters: the wavelength (λ) and the amplitude (A). In our study, we consider a parabolic profile (s/h=0.7) for the cross-section of the micro-grooves and we study two different sinusoidal riblets varying the amplitude (A=0.6mm and A=0.15mm), for a fixed value of the wavelength λ/s=64mm. Our load cell measurements show a consistent effect of the amplitude of the sinusoidal riblets on the friction drag reduction. In particular, while the longitudinal riblets feature drag reductions of the order of 7.7% at s/h=13 (in good agreement with Bechert et al. 1997), the sinusoidal riblets can achieve values as large as 10% for similar values of s/h. Stereoscopic-PIV measurements show the different near wall structure of the flow, when the three-dimensional riblets are employed.

1This work has been developed under a CS2-RIADP program.

8:13AM C26.00002 Streamwise evolution of drag reduced turbulent boundary layer with dilute polymer solutions, YASH SHAH, SERHIY YARUSEVYCH, University of Waterloo — The effect of dilute polymer solutions on the evolution of flat-plate turbulent boundary layer has been investigated experimentally. Three different injection concentrations of 100, 500 and 1000 ppm of polyethylene oxide (PEO) were injected through an inclined slot on the plate and particle image velocimetry measurements were performed to characterise boundary layer development and compare the results to the baseline cases without injection and with water injection. Drag reduction (DR) was found to decrease approximately linearly with injection distance for all injection concentrations. The results show that polymer injection has a significant effect on the flow statistics, including the extent of viscous sublayer and the characteristics of the log layer. The polymer injection also attenuates peak velocity fluctuations and moves them away from the wall with increasing polymer concentration. It is also shown that the ejection and sweep motions are weakened notably by polymer injection.

1The authors gratefully acknowledge the support of the Natural Sciences and Engineering Research Council of Canada (NSERC).
8:26AM C26.00003 Passive Actuation of Scales Modeled after Shark Scales to Delay Separation in a Steady Turbulent Boundary Layer1, CHASE PARSONS, AMY LANG, LEONARDO SANTOS, ANDREW BONACCI, SARAH FOLEY, The University of Alabama — Delaying the onset of flow separation is of great interest in the field of fluid mechanics to improve the overall aerodynamic efficiency of aircraft. This project seeks to investigate passive flow control using shortfin mako shark inspired manufactured scales in turbulent boundary layer separation. Previous studies have demonstrated the effectiveness of similar devices placed inside the separation bubble. In this study, the scales are placed in front of the separation point to investigate the effectiveness to delay separation. Reversing flow is the primary mechanism causing the actuation of the shark scales, so under these test conditions, it is hypothesized that reversing flow low speed streaks can actuate and be controlled by the scales, thus delaying the onset of separation. To generate a controlled adverse pressure gradient, a rotating cylinder induces separation at a chosen location within a flat plate turbulent boundary layer ranging from Re=495,000 to Re=710,000. With this thick boundary layer, DPIV is used to measure the flow characteristics. The goal is to better understand the fundamental mechanisms by which shark scales can induce passive flow control with the aim of fabricating surfaces suitable for real aircraft applications.

1US Army Grant W911NF1510556

8:39AM C26.00004 Flow mechanism at the interface layer of the bio-inspired coated surface in a turbulent channel flow , VENKATESH PULLETKURTHI, CARLO SCALO, LUCIANO CASTILLO, Purdue University — We have investigated the turbulent channel flow coated with bio-inspired surface via fully unstructured Pseudo-spectral method. The present study is inspired by the denticles present on the skin of the mako shark. Previous experimental studies by Bocanegra et. al (2018) revealed that the denticle like micro structures (85 μm) delayed the separation in an adverse pressure gradient flow. However, in spite of several studies to understand the flow near the wall, the interaction at the interface layer and inside the microstructures is not fully understood. Here, we use high resolution adaptive mesh refinement based DNS simulations to resolve the flow in the interface layer and inside the microstructures to shed light on the mechanism between the interface layer which is responsible for the drag reduction and/or delay in separation. This study also will help to model the interface layer boundary conditions for numerical modeling.

8:52AM C26.00005 Deviations in Polymer Drag Reduction Performance with Mechanical Degradation1, ZEESHAN SAEED, YASAMAN FARSIANI, DR. BRIAN ELBING, Oklahoma State University — Polymer drag reduction studies, although show great promise in flow control applications, are significantly limited by a problem; mechanical degradation of polymers in shear flows. Insights into this problem were made by comparing the drag reduction performance (slope increments on Prandtl-Karman (P-K) plots) of Polyethylene oxide (PEO) samples with and without degradation. The molecular weights of PEO samples—a measure of the extent of their degradation—were determined by matching the onset of drag reduction i.e. the intersection of the polymeric curve with the Prandtl-von Karman law on the P-K plots. Range of mean molecular weights (0.6 – 8 million g/mol) of PEO samples were included in the test matrix. Higher molecular weight samples were mechanically degraded to lower mean molecular weights that matched the molecular weights of available non-degraded samples (e.g. 4 million g/mol was degraded to 0.6 million g/mol). Comparisons of resulting slope increments determined from P-K plots of the degraded and non-degraded samples were then scaled with a function based on their mean molecular weights to show how polydispersity is central in determining the flow characteristics. This presentation reports the findings from the experiments and analysis mentioned above.


9:05AM C26.00006 Drag-reduction curves for anisotropic permeable substrates1, GARAZI GOMEZ-DE-SEGURA, RICARDO GARCIA-MAYORAL, University of Cambridge — We present DNSs of channel flows bounded by modelled streamwise-preferential permeable substrates. The resulting drag curves are similar to those of riblets. For small permeabilities, the curves exhibit a linear regime, where drag reduction is proportional to the difference between the streamwise and spanwise permeabilities. This breaks down for a critical value of the wall-normal permeability, beyond which spanwise-coherent, Kelvin-Helmholtz-like structures develop, and the performance begins to degrade. We present simple linearised models to predict both the linear regime and its breakdown, which yield accurate a priori estimates for the substrates’ performance. The largest drag reduction observed in our DNSs is ≈ 20 – 25% at a friction Reynolds number Reτ = 180, at least twice that obtained for riblets.

1Engineering and Physical Sciences Research Council (EPSRC)

9:18AM C26.00007 Role of wall-attached structures in frictional drag reduction by streamwise1, MIN YOON, HYUNG JIN SUNG, KAIST — The role of wall-attached structures in the frictional drag reduction by the Navier slip is explored. The wall-attached structures are extracted from the clusters of streamwise velocity fluctuations in a turbulent channel flow (Reτ = 470). The skin friction coefficient (Cf) is decreased by 35%. A dataset of the no-slip condition (Reτ = 577) is also included for comparison. The wall-attached structures extend toward the upstream in the vicinity of the wall by the slip. The convection velocity of the wall-attached structures increases near the wall, leading to the wide influence on the inner region via roll-cell motions. The vortical structures circumscribing the wall-attached structures are attenuated, since the mean shear of the structures is decreased by the slip. The contribution of the wall-attached structures to Cf is quantified through the skin friction decomposition, which can be derived from the mean vorticity equations. The advective vorticity transport and vortex stretching terms around the wall-attached structures are found to dominate the contributions to the frictional drag. The wall-attached structures are responsible for 53.2% of the total reduction of Cf.

1This work was supported by a grant from the National Research Foundation of Korea (NRF) (No. 2019022966 and 2019M3J1B7025091) and partially supported by the Supercomputing Center (KISTI).
The Baseball Seam: Clever and Capable Passive Flow Control
BARTON SMITH1, ANDREW SMITH2, Utah State University — It is obvious to any observer of baseball that the aerodynamics of the ball are important, both for pitched and batted balls. Much has been written about the well-known Magnus Effect, or the force on a moving ball due to its rotation. Less is known about forces due to the wake of the ball. Baseball seams make baseballs very interesting when compared to other sports balls. They play two roles: As many have speculated, when located in the favorable pressure gradient on the front of the ball, they can cause laminar flow to become turbulent, which subsequently modifies the wake of the ball. More surprisingly, when located in the adverse gradient on the back of the ball, they can also modify the location of boundary layer separation and can make the wake (and thus the force on the ball) asymmetric, leading to movement. In this talk, we will discuss these effects and the possibility of the existence of the “laminar express” 2-seam fastball that moves due an asymmetric wake rather than by Magnus effect.

1Professor, Mechanical and Aerospace Engineering
2Graduate Student, Mechanical and Aerospace Engineering

Passive control of vortex-induced vibration of a sphere,1, ANCHAL SAREEN2, University of Minnesota, JOHN SHERIDAN, KERRY HOURIGAN, MARK THOMPSON, Monash University — Although passive methods for controlling vortex-induced vibrations (VIV) are extensively studied for a circular cylinder, such methods remain unexplored for a basic three-dimensional bluff body, a sphere. In this study, we use a sphere trip wire as a passive method to control sphere VIV. The effect of a surface trip is experimentally investigated for varying diameter (1.25 x 10^-2 < k/d < 6.63 x 10^-2) and stream wise location (φ = 20° -70° from the stagnation point) of the trip wire for a wide range of reduced velocities (3 < U* < 20). It was found that the vibration amplitude decreases progressively with the increase in the stream wise location angle (φ) of the trip wire. The control was highly effective in mode II and mode III of the VIV response with maximum reduction of up to 97.8% for φ = 60°. Interestingly, thicker trip wires (k/d > 1.25 x 10^-2) were more effective in mode I, but showed a galloping response for higher reduced velocities.

1This work is supported by Australian Research Council Discovery Grants: DP150102879 and DP170100275
2Membership Pending

Underlying Mechanisms of Drag Reduction in Turbulent Flows1, ALEX ROGGE, JAE SUNG PARK, University of Nebraska - Lincoln — Turbulent flow control is of great importance in fundamentals and applications due to the potential benefits associated with it, particularly regarding drag reduction for energy savings. In this study, we will investigate three strategies to better understand their underlying mechanisms for drag reduction in turbulent channel flows. These strategies include using a spanwise body force, adding a small concentration of long-chain polymers into the fluid, and using a superhydrophobic surface on the channel walls. Direct numerical simulations were performed to elucidate the mechanisms at play. Analysis is based on the lifetime of turbulent phases represented by the active and hibernating phases of minimal channel turbulence (Xi and Graham PRL 2010). Given similar drag reduction percentages, the polymer and slip cases show similar mechanisms, while the body force case shows a different mechanism. The polymers and slip surfaces cause hibernating phases to happen more frequently, while the phase duration remains almost constant. The body forces prolong the duration of hibernating phases, while these phases become less frequent. Lastly, these drag reduction mechanisms and their various behaviors with respect to control parameters will be further discussed and analyzed.

1This work was supported in part by the National Science Foundation, Grant No. OIA-1832976.

Session C27 Biological Fluid Dynamics: Swimming, Sensing, Navigating

Development and validation of a bio-inspired, self-propelled metachronal swimming robot1, ARVIND SANTHANAKRISHNAN, MITCHELL FORD, Oklahoma State University — Metachronal swimming is a method of drag-based locomotion used by crustaceans such as krill, mysids, and shrimp. Studies of metachronal swimming can help in understanding ecologically important daily vertical migrations of these organisms and their hydrodynamic signaling mechanisms. We developed a robotic model ("krillbot") and validated its performance using published data on Pacific and Antarctic krill. Dynamic scaling was used to design the krillbot body and test conditions. The krillbot was suspended in an 8-foot long tank filled with water-glycerin mixture, and was allowed to self-propel on an air bearing. Time-resolved PIV measurements during self-propulsion showed that interaction of shear layers of adjacent paddles resulted in the formation of a continuous jet moving in the caudoventral direction. Swimming speed and orientation of the jet varied with phase lag and paddling frequency. Displacement efficiency and Reynolds number based on swimming speed were found to fall within the range observed in freely swimming krill.

1This work was supported by the National Science Foundation (CBET 1706762 and CBET 1512071).

Importance of appendage spacing in metachronal swimming1, MITCHELL FORD, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Drag-based metachronal paddling of multiple appendages is a common swimming strategy used by numerous aquatic animals (such as copepods, krill, and comb jellies) across a wide range of Reynolds numbers. These organisms have been reported to have a narrow range of dimensionless appendage spacing (ratio of inter-appendage spacing to appendage length) between 1/5 and 2/3 (Murphy et al., Mar. Biol. 158, 2011). Small inter-appendage spacing could allow for synergistic interaction of shear layers formed by paddling of adjacent appendages, whereas large inter-appendage spacing could effectively isolate individual appendages from each other. Using a robotic metachronal swimming model ("krillbot"), we investigated the effects of varying appendage spacing on propulsive forces, swimming speed, and wake characteristics for appendage spacings both within and greater than the biological range. Swimming speed was found to increase both with closer spacing of padding appendages and with increasing paddling frequency. PIV-based flow visualization results will be presented.

1This work was supported by the National Science Foundation (CBET 1706762 and CBET 1512071).
8:26AM C27.00003 Hydrodynamics of caridoid escape response in krill. ANGELICA CON-NOR, D. ADHIKARI, DEVESH RANJAN, D.R. WEBSTER, Georgia Tech — Krill are shrimp-like crustaceans and are a keystone species in many deep-water food webs. Due to their abundance and sensitivity to changes in the environment, there are many studies on krill ecology. But, there is limited quantitative analysis of the hydrodynamics of their locomotion and biomechanics. The length of Antarctic krill can range 2-6 cm, and these animals typically swim in a low to intermediate Reynolds number (Re) regime. The caridoid escape response, a maneuver unique to jumping arthropods, occurs when the minimal abdominal flexions resulting in powerful backward strokes. For the first time, the propulsion behavior and flow disturbance of a caridoid escape response performed by an Antarctic krill (Euphausia superba) has been quantified. A high-speed tomographic Particle Image Velocimetry (tomo-PIV) system quantifies three-dimensional flow fields around a free swimming E. superba and its wake. The specimen is roughly 3 cm in length and by using this tail flipping mechanism, it is able to accelerate backwards increasing its speed by 2 orders of magnitude in an interval of 0.025 s to a maximum speed of 25 cm/s. The data from these flow fields are used to calculate the changes in the velocity and vorticity field shedding light on both the flow behavior in this Re regime and intricacies of the bio-locomotion of zooplankton.

8:39AM C27.00004 Can the copepod seta sense the hydrodynamic disturbance of prey entrained in the feeding current? XINHUI SHEN, MARCOS MARCOS1. Nanyang Technological University, HENRY FU, University of Utah — The prey detection of feeding-current feeding copepods is achieved by beating their cephalic appendages to generate flow entrainment and utilizing their mechanoreceptive setae to sense the presence of the prey. The hydrodynamic characteristics of the copepod’s feeding current have been extensively studied; however, there is little knowledge on if the copepod seta is capable of sensing the hydrodynamic disturbance of prey, or otherwise a direct contact of the setae and prey is required. Here we present a mechanical model to examine the deformation mechanics of the copepod setae when subjected to the flow disturbance of an inert particle entrained in the copepod’s feeding current. We first determine the hydrodynamic characteristics of a copepod and beating stroke of its cephalic appendages through video analysis, and utilize the immersed boundary method to solve for the flow fields around the seta with and without the presence of the entrained prey. We then proceed to evaluate the setal deformation induced by such flows, and demonstrate that the flow disturbance induced by the entrained prey leads to a different setal deformation pattern, which may be sensed by the copepod.

1This author’s full name is Marcos

8:52AM C27.00005 Non-axisymmetric flow and sensing around copepods. JULIAN HACHEMEISTER, DAISUKE TAKAGI, University of Hawaii at Manoa — Microscopic organisms generate a variety of viscous flow fields which are critical for locomotion, feeding, and sensing. We report on a simplified model of the flow field generated by a feeding copepod while sinking, swimming and hovering. In each case, we compare the different flow fields and compute the strength of the hydromechanical signal due to any suspended particle. Among the three modes, hovering is most effective in bringing in a fresh supply of nutrients from its surroundings. Our study shows an approximate range in which a copepod may sense its prey and the role mechanical sensing plays in feeding.

9:05AM C27.00006 Numerical investigation of seal whiskers in distinguishing the patterns of wakes induced by moving objects with different shapes. GENG LIU, QIAN XUE, XUDONG ZHENG, University of Maine — Phocid seals are able to use their whiskers (or vibrissae) to detect and track artificial and biogenic hydrodynamic trails. Well trained seals are even able to discriminate the size and shape of upstream moving objects through their wakes. The present study employs a one-way coupling model of flow-structure interaction (FSI) to investigate the hydrodynamic mechanism of the wake discrimination. The root mechanical signals of whisker arrays in the wakes induced by different moving objects are simulated and analyzed in detail. It is found that the signal patterns of whisker arrays are associated with the strength and the direction of the jets induced by the 3D vortices in the wake. The distinct signal patterns enable the seal to discriminate the shapes of upstream moving objects. In addition, a theoretical model is built to decoding the relationship between the location of the disturbance source and the sensed flow information.

9:18AM C27.00007 A 3D computational fluid dynamics study of the swimming of the larva of mosquito (Chironomus plumosus). BOWEN JIN, Beijing Computational Science Research Center, HAOXIAO LUO, Vanderbilt University, YANG DING, Beijing Computational Science Research Center — The larva of Chironomus plumosus has a cylindrical body with a length about 14mm. Experiments have shown that it swims by periodically bending its body into a circle (the head and tail nearly in touch) and then unfolding it. However, the propulsion mechanism of the larva is not well understood. Here we use 3D computational fluid dynamics to simulate the swimming of the larva. According to the experimental observations of the movement pattern, the centerline curvature $k$ is prescribed in the form of a sinusoid function. The rotational and translational velocities are obtained by coupling the body with the fluid. The simulation results show that the greatest force and thrust are generated on the head and tail during the unfolding stage. By adjusting the time fraction $\gamma$ of the unfolding stage, we find that both the swimming speed and the energetic efficiency increase with decreasing $\gamma$. However, the difference in swimming speed is only significant at intermediate Reynolds number (Re=1000). When Reynolds number increases (Re=3000) or decreases (Re=30), the difference in speed becomes smaller. Our study suggests that the kinematics of the larva of mosquito is specialized for swimming at intermediate Reynolds numbers.

9:31AM C27.00008 Elastic hoops jumping on water: drag-dominant model of water-jumping arthropods. HAN BI JEONG, Seoul National University, EUNJIN YANG, Korea Institute of Science and Technology Evaluation of Planning, YUNSUK JEUNG, Seoul National University, JULIETTE AMAUGER, Ecole Normale Superieure, HO-YOUNG KIM, Seoul National University — Some remarkable milli-scale organisms jump from the water surface using surface tension, such as the well-studied water strider. Larger arthropods, however, require a greater force to complete the same task. In the case of fishing spiders, a pressure drag, rather than capillary forces, is utilized to propel the spiders’ bodies into a successful jump. Such a strategy is also discriminated from the unsteady added inertia effects employed by a basilisk lizard, a water-walking reptile. Here, we present a mathematical model of thin elastic hoops jumping on water, inspired by the fishing spider. A pre-deformed hoop coated with superhydrophobic particles floating on water shows similar dynamic conditions to that of the jumping mechanics of fishing spiders. When released, the water applies a force against the deforming hoop, dominantly in the form of drag, propelling the hoop into the air. By combining the vibration model of the elastic hoop and the time-varying drag forces induced by fluid motion, we accurately predict the trajectory and jump efficiency of the hoops. This work can be used to develop large-scale water-jumping robots and to understand the water-jumping mechanics of large semi-aquatic arthropods.
9:44AM C27.00009 Swimming and settlement of coral larvae on structured surfaces in unsteady shear flow , DANIEL GYSBERS, MARK LEVENSTEIN, GABRIEL JUAREZ, University of Illinois at Urbana-Champaign — The large and amazing structures that we know as coral reefs have humble beginnings as tiny (<1 mm) swimming organisms. Coral larvae must navigate the vast marine environment to locate a suitable surface where they will permanently settle on by responding to various chemical, biological, and physical cues. We present experimental results of coral larval swimming and settlement on varying structured surfaces in unsteady shear flow. Our experiments use PTV of swimming larvae and PIV of unsteady shear flow in a U-shaped oscillatory flume to investigate the effect of hydrodynamic interactions between coral larvae and local flow structures generated by surface topology on settlement rates.

9:57AM C27.00010 Role of large-scale advection and small-scale turbulence on the vertical migration of gyrotactic swimmers , CRISTIAN MARCHIOLI, University of Udine, GAETANO SARDINA, Chalmers University of Technology, LUCA BRANDT, Linn FLOW Centre and SeRC, KTH Mechanics, ALFREDO SOLDATI, TU Wien — We use DNS-based Eulerian-Lagrangian simulations to investigate the dynamics of small gyrotactic swimmers in free-surface turbulence. Swimmers are characterized by different vertical stability: some realign with a characteristic time smaller than the Kolmogorov time scale, $\tau_K$, while others possess a re-orientation time longer than $\tau_K$. We cover one order of magnitude in the flow Reynolds number, and two orders of magnitude in the stability number, which measures bottom heaviness. We observe that large-scale advection dominates vertical motion when the stability number, scaled on the local Kolmogorov time scale, is above unity: This leads to enhanced migration towards the surface, particularly at low Reynolds number, when swimmers can rise through surface renewal motions that originate from the bottom-boundary turbulent bursts. Small-scale effects become important when the Kolmogorov-based stability number is below unity: Migration towards the surface is hindered, particularly at high Reynolds, when bottom-boundary bursts are less effective in bringing bulk fluid to the surface. We demonstrate that a Kolmogorov-based stability number around unity represents a threshold beyond which swimmer capability to reach the surface and form clusters saturates.

Sunday, November 24, 2019 8:00AM - 10:10AM –
Session C29 Biological Fluid Dynamics : Phonation and Speech 611 - Michael Krane, Penn State University

8:00AM C29.00001 Control volume analysis of phonatory aerodynamics using velocity and pressure measurements1 , TIMOTHY WEI, University of Nebraska, HUNTER RINGENBERG, University of Colorado - Boulder, NATHANIEL WEI, Stanford University, DYLAN ROGERS, University of Nebraska, FEIMI YU, LUCY ZHANG, Rensselaer Polytechnic Institute, MICHAEL KRANE, Penn State Applied Research Lab — In a collaborative effort, flow through simplified models of human vocal folds are being examined both experimentally and computationally. The experimental work described in this talk entails a 10x scaled-up model in a free surface water tunnel. In addition to the 10x physical scale, the reduced kinematic viscosity of water allows for a 1500x reduction in frequency to match the Reynolds numbers and reduced frequencies of human phonation. As such DPIV measurements, coupled with time resolved pressure measurements along the vocal fold model, have sufficiently high spatial and temporal resolution to accurately study the dynamics and energetics of the underlying fluid dynamics. This, in turn, provides direct linkage with aerodynamics analyses done using a matching computational model. In this study, we use integral control volume analysis in both the scaled-up physical experiment and the computational model to examine the momentum and energy balance in the flow. Specific analysis of the Bernoulli equation for a range of Reynolds numbers and reduced frequencies will be examined. Comparison with computational results and observations on the key phenomena related to phonation will be provided.

1Support from NIH R01DC005642 is appreciated.

8:13AM C29.00002 Vocal Fold Trauma: Deleterious Compensatory Behaviors in Response to Benign Lesions1 , MOHSEN MOTIE-SHIRAZI, BYRON ERATH, Clarkson University — Abnormally high vocal fold (VF) contact pressures result in the formation of benign lesions such as polyps and nodules, which may produce highly asymmetric VF motion, disrupt glottal aerodynamics, and prevent VF closure. In response, compensatory behaviors, such as increasing the subglottal pressure, are often performed to maintain acoustic output. It is hypothesized that increased subglottal pressure leads to a deleterious cycle of higher contact pressure, and subsequently, additional VF trauma. The objective of this work is to quantify VF contact pressures during this common compensatory behavior. Four-layer synthetic VF models are fabricated and investigated in a hemilaryngeal configuration with the wall contact pressure measured at the midpoint of the VF contact zone. Three sizes of lesions are modeled by inserting a spherical ball in the mid anterior-posterior direction of the superficial lamina propria, below the epithelium. Contact pressure and radiated acoustic sound pressure level (SPL) are first measured in normal models with no lesion and then compared to those with a lesion, where the subglottal pressure is adjusted to match the radiated SPL between the two cases. Results demonstrate that localized lesion stiffness and compensations via subglottal pressure significantly alter the VF kinematics, producing much higher contact pressures that increase with lesion size.

1This research was funded by the National Institutes of Health (NIH) National Institute on Deafness and other Communication Disorders Grant P50 DC015446.

8:26AM C29.00003 Reduced-order glottal airflow model enhanced by machine learning , ZHENG LI, YE CHEN, HAOXIANG LUO, Vanderbilt University — Complementary to expensive 3D flow simulations, reduced-order glottal airflow models are useful in the simulation of vocal fold vibration for various purposes such as tissue property identification and optimization of surgical implants. Existing reduced-order models are typically based on the Bernoulli principle and have limited accuracy. In recent works, we have developed a novel one-dimensional flow model including pressure loss along the glottis and also the entrance effect. The model has shown advantages over the Bernoulli based model. In this work, we introduce machine learning to enhance this one-dimensional flow model. In particular, we firstly perform 3D fluid-structure interaction (FSI) simulation for vocal fold vibrations with different vocal fold stiffnesses and medial thicknesses. Using the 3D data, sparse regression is performed to estimate the loss coefficient and entrance effect parameter for the reduced-order flow model. We then combine this enhanced reduced-order flow model with 3D FEM tissue model to simulate the vocal fold vibration. In comparison with full 3D FSI model, very good agreements are achieved in terms of vibration amplitude, frequency, as well as phase delay of the medial surface in all cases considered.
8:39AM C29.00004 Acoustic coupling effect on glottal dynamics during phonation.1, DARUISH BODAGHI, WEILI JIANG, QIAN XUE, XUDONG ZHENG, University of Maine — In this study, a two-dimensional flow-structure-acoustics interaction computational model was utilized to examine the effect of the acoustic coupling on glottal dynamics during normal human phonation. An incompressible Navier-Stokes equation based flow solver, three mass based vocal fold model and a hydro/acoustic splitting method based acoustics solver were coupled for simulating the three-way interaction. The hydro/acoustic splitting method is first verified for low Mach number wall bounded flow against compressible N-S solutions obtained by using Fluent. Two simulation cases, with and without the acoustic coupling, were studied to identify the role of the acoustic coupling in normal phonation. The results indicate that, while the incompressible flow model could roughly capture basic glottal dynamics, the acoustic coupling has noticeable effects on both glottal flow and vocal fold dynamics.

1Acknowledge NIH R01DC005642

8:52AM C29.00005 A framework for simulation of sibilant fricatives using implicit compressible flow solver. HSUEHJUI LU, CHUNGANG LI, Kobe University, AKIYOSHI IIDA, TSUKASA YOSHINAGA, Toyohashi University of Technology, KAZUNORI NOZAKI, Osaka University, MAKOTO TSUBOKURA, Kobe University — A numerical framework for modeling the sibilant fricative production is built in this study. The implicit time scheme with immersed boundary method based on a hierarchical structure grid as well as the modified solution-limited time stepping method for the compressible flow are adopted. Firstly, the acoustic resonance generated from the flow around the single plate is simulated to validate the numerical scheme and the result shows that this framework is highly efficient and suitable for the massive parallelization system to tremendously save the calculation time. Then, the simulation for a simplified model of the sibilant /s/ is conducted and SPL profiles are in good agreement with the experimental results. Finally, the simulation for a realistic geometry of sibilant /s/ scanned from the human vocal tract is performed to demonstrate that this framework is capable of making a contribution to the dental treatment in the near future.

9:05AM C29.00006 Phonation energy budget computed from high-fidelity aeroelastic-aeroacoustic simulation1, LUCY ZHANG, FEIMI YU, Rensselaer Polytechnic Institute, MICHAEL KRANE, ARL, Pennsylvania State University — A rigorous accounting of phonation energy utilization is presented, using high-fidelity computer simulations. The simulation uses the modified immersed Finite Element (iFEM) formulation, supplemented by boundary condition control using Perfectly Matched Layers. Vocal folds mimic the swept-ellipse multilayer rubber model used in coordinated experiments. Simulations were run for a range of subglottal pressures. Simulation results are used to compute terms of the integral energy equation for the volume containing air in the larynx. Flow work terms in particular are decomposed to clarify power transfer mechanisms. The mechanism described by each energy equation term is then classified in terms of its role (input, output, loss). Laryngeal acoustic efficiency is also presented.

1NIH R01DC005642

9:18AM C29.00007 Phonation aeroacoustic sources identified from simulation1, FEIMI YU, LUCY ZHANG, Rensselaer Polytechnic Institute, MICHAEL KRANE, Applied Research Laboratory, Penn State University — The principal aeroacoustic sources in phonation are estimated from a high-fidelity computer simulation. The simulation uses an immersed Finite Element (iFEM) formulation, supplemented by boundary condition control using Perfectly Matched Layers. Vocal folds mimic the swept-ellipse multilayer rubber model used in coordinated experiments. Simulations were run for a range of subglottal pressures. For each simulation, the principal aeroacoustic sources were deduced: a volume source due to changes in vocal fold volume, and a dipole source associated with vocal fold drag. Recent arguments regarding the equivalence of vocal fold drag and transglottal pressure force, and the relationship between vocal fold drag and glottal volume flow are also evaluated.

1NIH R01DC005642

9:31AM C29.00008 2. Simultaneous measurements of vocal fold wall motion, aerodynamic and acoustic pressure, and volume flow1, FAITH BECK, PAUL TRZCINSKI, ADAM NICKELS, ZACHARY YOAS, JEFF HARRIS, MICHAEL KRANE, Applied Research Laboratory, Penn State University — Simultaneous measurements of acoustic pressure, transglottal pressure, volume flow, and vocal fold surface motion are reported. Measurements were conducted in a hemilarynx configuration consist of the Penn State Upper Airway Model (PSUAM), using a multilayer swept-ellipse vocal fold model. Pressure and volume flow measurements were conducted as described in other studies in the PSUAM, but vocal fold surface motion was also acquired by imaging the surface with three Vision Research v1212 high-speed cameras. The surface motion was estimated from the video using DaVis Strainmaster. Vocal fold motion is decomposed using POD and correlated to pressure and flow measurements.

1Acknowledge NIH R01DC005642 and Penn State ARL (IRAD)

9:44AM C29.00009 Vocal fold asymmetry effects on phonation aeroacoustic source strengths1, PAUL TRZCINSKI, ZACHARY YOAS, MICHAEL KRANE, Applied Research Laboratory, Penn State University — Measurements of aeroacoustic source strengths in a physical model of the human airway are taken for variations in prephonatory mechanical symmetry. Opposing synthetic silicone vocal folds were placed in the Penn State Upper Airway Model (PSUAM) and subjected to various driving pressures for which fold oscillations occur. Vocal fold symmetry was varied by changing the multilayer structure between the two vocal fold models. For each vocal fold pair, measurements include unsteady transglottal pressure, acoustic pressure in the vocal tract and trachea regions, projected glottal area, and time-varying volume flow. In particular, transglottal pressure has been shown to be an accurate measure of vocal fold drag, the aeroacoustic source strength. Measuring vocal fold drag indicates how asymmetry affects phonatory sound production.

1Acknowledge NIH R01DC005642
9:57AM C29.00010 4. Phonation energy utilization and vocal efficiency estimated from measurements in a physical model of the human vocal system¹, MICHAEL KRANE, PAUL TRZCINSKI, ZACHARY YOAS, Applied Research Laboratory, Penn State University — Measurements performed in the Penn State Upper Airway Model of acoustic pressure, transglottal pressure, volume flow, and glottal area are presented. These are used to estimate flow work terms of the integral energy equation, applied to the vocal system, to identify principal power transfers during phonation. These terms are estimated for the larynx, vocal tract, and complete system. Each power flow is then classified as an input, output, or loss mechanism, according to the sign of the average power transfer per cycle. Efficiencies for each subsystem, and the system as a whole, are then presented as the ratio of output to input power per cycle.

¹Acknowledgment NIH R01DC005642

Sunday, November 24, 2019 8:00AM - 10:10AM —
Session C30 Biological Fluid Dynamics : Cardiovascular Flows  612 - Jian-Xun Wang, University of Notre Dame

8:00AM C30.00001 Multiscale modeling of the cardiovascular deconditioning during spaceflight, STEFANIA SCARSOGLIO, CATERINA GALLO, LUCA RIDOLFI, Politecnico di Torino, Torino, Italy — Permanence in microgravity characterizing long-term spaceflights causes a general deconditioning of the cardiovascular system. A constellation of hemodynamic mechanisms - such as fluid shift, blood volume reduction, vessel elasticity changes, and cardiac atrophy - concurs to define a 0G adaptation point, which is identified by an overall relaxation of the cardiovascular system. This scenario promotes orthostatic intolerance on reentry and poses open questions on re-adaptation for spaceflights beyond 1 year. We here present a computational approach to compare the cardiovascular response in supine position on Earth and at 0G adaptation point during spaceflight. The proposed aim is twofold: (i) understand the underlying mechanisms leading to cardiovascular deconditioning; (ii) describe the hemodynamics of districts for which clinical data are not always feasible and accurate even on Earth. The present approach relies on a validated multiscale modeling of the cardiovascular system combining a 1D description of the arterial tree together with a lumped parameterization of the remaining regions (i.e., venous return, heart chambers, pulmonary circulation, baroreceptor regulation).

8:13AM C30.00002 Rich Dynamic Behaviors of Self-excited Oscillation of Collapsible Channel¹, QIUXIANG HUANG, FANG-BAO TIAN, JOHN YOUNG, JOSEPH C. S. LAI, The University of New South Wales, FLOW SCIENCE LAB TEAM — Fluid-structure interaction (FSI) in collapsible channel flow is numerically studied with an immersed boundary-lattice Boltzmann method. Compared with previous studies, current method is able to simulate nonlinear fully coupled FSI in two-sided collapsible channel and high Reynolds numbers flow (Re up to 2000). The stability of the hydrodynamic flow and collapsible channel walls are examined for a wide range of Reynolds numbers, structure-to-fluid mass ratio, external pressure and wall thickness. Based on the numerical simulations, we (i) explore the physical mechanisms responsible for the onset of self-excited oscillations, and (ii) characterise the chaotic behavior of the collapsible channel walls. Rich dynamic behaviors of self-excited oscillation are observed. Regarding point (i), we identify that the flow bifurcate to bistable mode at Re=320 due to the symmetry breaking as the increase of Reynolds number. Besides, the external pressure applied on the elastic beams plays an important role in triggering the self-excited oscillation of the beam. And then for point (ii), the existence of chaotic behavior of the collapsible channel walls is confirmed by a very positive dominant Lyapunov exponent and the chaotic attractor in the velocity-displacement phase portrait.

¹Qiuxiang acknowledges the support of the University International Postgraduate Award by the University of New South Wales. The computation work of this research was partially performed on the National Computational Infrastructure supported by Australian Government.

8:26AM C30.00003 Optogenetics-Biofluid Model for Engineered Beating Cardiomyocytes using Meshfree Method, YASSER ABOELKASSEM, University of California San Diego — Optogenetic techniques make use of genetically encoded, light sensitive ion channels to manipulate cellular function with light. More recently, microbial opsins such as the light-gated ion channel channelrhodopsin-2 (ChR2) have been transfected into cardiomyocytes, allowing cardiac muscle contractions to be initiated by pulses of light. Cardiac optogenetics raises numerous interesting possibilities, including optical pacemakers, defibrillators, and flow pumping assistant devices accomplished through the precise spatiotemporal application of light excitations. In this study, an optogenetics-fluid mathematical model is proposed to study the flow motions induced by a single engineered cardiac cell. The optogenetics module is based on Monte Carlo type of simulations to control light stimulus events, which are then used to probe the cellular wall contractions. The fluid module is derived based on a two-dimensional meshfree-Stokeslets computational framework. The results show that, cells with a slightly different beating profile can induce different flow field that is characterized by coherent vortices with different strengths and core sizes. This implies that, each cell induces unique flow biomarker “signature”, which can be used to better understand the intrinsic sub-cellular excitation-contraction processes of cardiac cells.

8:39AM C30.00004 A Multi-fidelity Ensemble Kalman Method for Inverse Problems in Cardiovascular Flows, HAN GAO, JIAN-XUN WANG, University of Notre Dame — In cardiovascular modeling, parameters associated with boundary conditions and mechanical properties are often unknown or uncertain, which can be calibrated using indirect and/or sparse clinical measurements based on data assimilation (DA) techniques. The ensemble Kalman filter (EnKF), as a derivative-free DA approach, has started to gain popularity for solving inverse problems in physiological modeling. However, the computational cost of the EnKF could be considerably high due to a large ensemble of costly forward simulations, in particular when the iterative Kalman updates are needed for nonlinear inversion (i.e., iterative ensemble Kalman method). In this work, we propose an efficient multi-fidelity ensemble Kalman inversion approach, where both the low- and high-fidelity forward models are utilized to accelerate the statistical convergence. Moreover, to improve the identifiability of the parameters to be inferred, additional physical/physiological constraints will be imposed by re-weighting the ensemble members in a Bayesian manner. Numerical examples of vascular flows in patient-specific geometries are presented to demonstrate the effectiveness and merit of the proposed framework.
8:52AM C30.00005 Super-resolution and Denoising of Flow MRI Data using Physics-Constrained Deep Learning , LUNING SUN, JIAN-XUN WANG, University of Notre Dame — The recent advances in the flow magnetic resonance (MR) imaging enable noninvasive and in vivo measurement of the blood flow velocity field. However, the resolution and signal-to-noise ratio (SNR) of flow MR images still remain the limiting factors in clinical practice. This is especially true when investigating small vascular structures such as intracranial aneurysms or treating near-wall regions where wall shear stress is calculated. Therefore, super-resolution and denoising of flow fields from MR images are of great importance and remain active research areas. In this work, we propose an innovative deep learning framework to upscale low-resolution flow fields and to reduce the measurement noise using physics-constrained deep neural networks (DNN). Specifically, a generative DNN will be trained on the low-resolution data to capture the flow field. In the meantime, the violation of physical laws will be penalized on a large number of spatiotemporal points where measurements are not available and noises are expected to be reduced. The trained generative model can reconstruct the flow field with arbitrarily high resolution. Several test cases with synthetic vascular flow data are studied to demonstrate the merit of the proposed method.

9:05AM C30.00006 Comparison of Multi-scale Models for Blood Flow in Zebrafish Brain1, MINGLANG YIN, Center for Biomedical Engineering, Brown University, USA, XIAONING ZHENG, Division of Applied Mathematics, Brown University, USA, ANSEL BLUMERS, Department of Physics, Brown University, USA, HIROYUKI NAKAJIMA, Department of Cell Biology, National Cerebral and Cardiovascular Center Research Institute, Japan, YOSUKE HASEGAWA, Institute of Industrial Science, The University of Tokyo, Japan, GEORGE KARNIADAKIS, Division of Applied Mathematics, Brown University, USA — The contribution of hemodynamics in developing zebrafish vasculature has long been recognized as one of the main factors in the mechanisms of vessel pruning. Using the modern computational fluid dynamics models, such as the three-dimensional (3D) Navier-Stokes model, the one-dimensional (1D) blood flow model, or the Dissipative Particle Dynamics (DPD), we performed the first detailed simulations to investigate the hemodynamics in zebrafish hindbrain. The simulations were performed on the same zebrafish hindbrain vasculature with the same Dirichlet boundary condition at its inlet. The flow rate and pressure profiles at outlets and inner points show a good agreement between the 1D and the other two models. This validates the 1D model accuracy in simulating blood flow at low Reynolds. Further investigations on non-Newtonian effect are ongoing. The performance of the 1D model facilitates its applications to further investigations on transport properties in physiological processes such as angiogenesis in zebrafish vasculature, mouse retinal plexus, or even a tumor.

9:18AM C30.00007 From whole-organ imaging to in-silico blood flow modeling: a new multi-scale network analysis for revisiting tissue functional anatomy1, FRANCK PLOURABOUE, Fluid Mechanics Institute of Toulouse UMR CNRS-UPS-INPT, J. COLOMBELLI COLLABORATION, C. BARreau, C. Guissard, L. Teysedre, J. Rouquette, A. Lorsignol, Louis Castella Collaboration, P. Kennel, J. Dichamp Team — We present a multi-disciplinary image-based blood flow perfusion modeling of a whole organ vascular network for analyzing both its structural and functional properties. We show how the use of Light-Sheet Fluorescence Microscopy (LSFM) permits whole organ micro-vascular imaging, analysis and modelling. By using adapted image post-treatments workflow, we could segment, vectorize and reconstruct the entire micro-vascular network composed of 1.7 millions vessels, from the tissue-scale, inside a ∼ 25 × 5 × 1 = 125 mm³ volume of mouse fat pad, hundred time larger than previous studies, down to the cellular scale at micron resolution, with the entire blood perfusion is modeled. Adapted network analysis revealed the structural and functional organization of mouse-scale tissue as strongly connected communities of vessels. These communities share out a distinct heterogeneous core region and a more homogeneous peripheral region, consistently with known biological functions of fat tissue. Graph clustering analysis also revealed two distinct robust meso-scale typical sizes (from 10 to several hundred times the cellular size), revealing, for the first time, strongly connected functional vascular communities. These communities networks support the reduced-order model can be constructed directly through the Jacobian matrix in the CFD solver. Reasonable accuracy and computational efficiency are demonstrated by comparing the reduced-order model result with the reference CFD result.

9:31AM C30.00008 Reduced-order modeling of flow in the circulatory system, CHENWEI MENG, MAHDI ESMAILY, Sibley school of mechanical and aerospace engineering, Cornell University — Lumpped parameter networks (LPN) coupled with 3D CFD simulations provide a mean to simulate the flow in the circulatory system at a reasonabe accuracy and affordable cost. In this method, the flow in major vessels is fully resolved, whereas it is modeled in the rest of the circulatory system using lumped components. Despite these attractive properties, the usage of 3D-LPN coupling method is hindered by need for reverse-engineering the LPN components through a tuning process, which itself requires performing many CFD simulations. To reduce the cost of such expensive calculations, a reduced-order model is introduced to replace the 3D model that is orders of magnitude less expensive to solve. The introduced reduced-order model is constructed using a circuit network analogy that can capture flow inertial and viscous losses in the 3D fluid domain. Additionally, we show that the reduced-order model can be constructed directly through the Jacobian matrix in the CFD solver. Reasonable accuracy and computational efficiency are demonstrated by comparing the reduced-order model result with the reference CFD result.

9:44AM C30.00009 Characterization of Flow Mediated Dilation via a Physics-Based Model, BCHARA SIDNAWI, Villanova University, ZHEN CHEN, CHANDRA SEHGAL, University of Pennsylvania, SRIDHAR SANTHANAM, QIANHONG WU, Villanova University, CELLULAR BIOMECHANICS AND SPORTS SCIENCE LABORATORY TEAM, DEPARTMENT OF RADIOLOGY AT THE UNIVERSITY OF PENNSYLVANIA TEAM — In this work, a preliminary physics-based model describing the transient behavior of the brachial artery during the Flow Mediated Dilation (FMD) test, is developed. Experimental diameter over time data were collected, via in-vivo ultrasound imaging. The model, which also accounts for mechano-transduction, was able to capture a key feature of the experimentally observed responses which a conventional viscoelastic model fails to explain. Characteristic dimensionless groups quantifying the physical state of the artery emerged from the model. The values of these dimensionless quantities, that predicted a response that best matched the experimental counterpart to a specific artery, were considered the values characterizing it. The meaning of these parameters and how they can be related to the cardiovascular health are discussed and explained. The transient physics manifested in the two-way Fluid-Structure Interaction (FSI) driving the FMD process, present an interesting opportunity to explore the nature of living materials making up the arterial walls, which would in turn lead to a better understanding and therefore detection of the onset of some forms of Cardiovascular Disease (CVD).
9:57AM C30.00010 Towards Blood Flow Velocimetry with X-Ray CT. BRENDAN COLVERT, ERIC YU, FRANCISCO CONTIJOCHE, ELLIOT MCEVIEGH, University of California San Diego — Cardiovascular disease (CVD) is a tremendous burden in terms of morbidity, mortality, and costs to the healthcare system. Various forms of CVD including atherosclerosis, valve and ventricular dysfunction, aneurysms, and thrombogenesis are associated with localized blood flow abnormalities. Accordingly, the ability to noninvasively interrogate physiological flows enables identification and diagnosis of disease, monitoring of therapies, and research on the hemodynamics of CVD. In the clinic, blood flow measurements are primarily made using phase contrast magnetic resonance imaging (PC-MRI) and ultrasonic color Doppler imaging. Certain limitations of these techniques for patients who have contraindications or suffer from arrhythmias, as well as the desire for volumetric flow information necessitate the development of a new modality for blood flow velocimetry. In this work, we propose a strategy to optimally integrate imaging data from contrast-enhanced X-ray CT scans with flow solvers. We evaluate the effectiveness of this strategy in the context of a simplified flow model. Our initial findings provide insight into the theoretical foundations of the proposed technique and lay the groundwork for further research on the use of X-ray CT for blood flow velocimetry.

Sunday, November 24, 2019 8:00AM - 10:10AM – Session C31 Biological Fluid Dynamics: Active Fluids

8:00AM C31.00001 Transport control for apolar active suspension in a rectangular channel using circular cylinders. SHENG CHEN, TONG GAO, MICHIGAN STATE UNIVERSITY, COMPLEX FLUIDS GROUP AT MSU TEAM — We study the control for transport of a dilute apolar active suspension in a rectangular channel using circular cylinders. We adopt a coarse-grained active liquid crystal model to describe collective unstable dynamics of non-motile but mobile rod-like particles, i.e., the “Extensors”, in Stokes flows. By using a Galerkin mixed finite element method, we previously revealed various patterns of spontaneous coherent flows that can be unidirectional, traveling-wave, and chaotic for the case without cylinders inside. To uncover the transport control using circular cylinders, we first study how fixed cylinders change the flow transitions in channel by systematically changing the cylinder size, separation distance, and the channel width. We further study if the flow transitions alter when the cylinders are free to rotate about a fixed axis.

8:13AM C31.00002 Chaotic mixing of an active nematic in viscous environments. AMANDA TAN, KEVIN MITCHELL, LINDA HIRST, University of California, Merced — The unifying theme of active matter is that collections of subunits consume energy locally, translate that energy into movement, and ultimately produce large-scale flows. Such materials are out of equilibrium systems. A fascinating example of a tunable non-equilibrium system is the microtubule/kinesin-based active nematic that forms at an oil/water interface. This material, based on biological polymers and molecular machines, produces a steady-state in which we observe moving topological defects that braid around one another. Defects are continuously created and annihilate, generating fluid flows. A direction of interest in non-equilibrium systems is to study how the active material responds to changes in its environment. To generate external environmental changes, we change the viscosity of the oil that the network is in contact with and probe subsequent changes in the morphology and speed of the network. Lower viscosity oils give rise to higher defect densities, lower defect densities, and vice versa for higher viscosity oils. Using analysis from chaotic advection theory, we explore how environmental changes affect various quantitative parameters such as the local fluid stretching rates within the network and the topological entropy as calculated from defect braiding.

8:26AM C31.00003 Stress relaxation in active suspensions. PRABHU NOTT, Indian Institute of Science, SANKALP NAMBIAR, Jawaharlal Nehru Centre for Advanced Scientific Research, PHANI KANTH SANAGAVARAPU, Indian Institute of Science, GANESH SUBRAMANIAN, Jawaharlal Nehru Centre for Advanced Scientific Research — The rheology of ‘active’ suspensions has been a subject of considerable interest in recent years, but mostly under steady shear. In a recent experimental study [1], the response of a dilute suspension of E. coli was characterized for impulsive initiation and cessation of shear. Interesting features in the relaxation of the shear viscosity were found, as the shear rate was varied, including regimes of apparent superfluidity. Here, we report a theoretical study that attempts to explain and analyze the observed rheological response [2]. Starting from a kinetic equation appropriate for rod-shaped bacteria like E. coli, we determine the evolution of the orientation distribution as a function of time following a step change in the shear rate, and thereby, the temporal evolution of the shear viscosity. One of the most striking aspects of such properties is the recent experimental observation of a vanishingly small shear viscosity in suspensions of swimming bacteria. Here, we present our recent results on the connection between this phenomenon and the onset of collective motion in bacterial suspensions. We find that confinement strongly influences both phenomena, and that the apparent viscosity drops to zero before the transition to collective motion. We compare our predictions against the active gel theory, and discuss their relevance to recent experiments.

1National Institutes of Health (NIH) K01 HL 143113-01 to F.J.C. and R01 HL 144678 to E.R.M. UCSD Institute of Engineering in Medicine (IEM) 2017 Galvanizing Engineering in Medicine (GEM) Award to E.R.M.

8:39AM C31.00004 Self-propulsion of active colloids via ion production. MARCO DE CORATO, XAVIER ARQU, TANIA PATIO, Institute for Bioengineering of Catalonia (IBEC), MA ARROYO, LaCN, Universitat Politècnica de Catalunya, SAMUEL SNCHEZ, Institute for Bioengineering of Catalonia (IBEC), IGNACIO PAGONABARRA, Universitat de Barcelona — Active particles that harness chemical energy from the environment and turn it into directed motion attracted great interest in the recent years. Several applications have been envisaged for these particles from pollutant removal to anti-cancer therapies. To optimize active colloids for advanced applications, one has to achieve fundamental understanding of their dynamics in fluidic environments. Here, we develop a model for the self-propulsion of a chemically active colloid that asymmetrically releases ionic species. By solving the relevant equations using simulations and a perturbation expansion, we evaluate the velocity of the active particle as a function of the main parameters. Our results highlight several novel aspects that are qualitatively different from other mechanisms. The active particle can reverse direction of motion by changing the salt concentration in the solution and can propel even if it is not charged. We find an optimal condition for self-propulsion and a novel regime in which the velocity is independent of the ionic strength of the environment. The model quantitatively captures the salt-dependent velocity measured in experiments using active colloids that propel by decomposing urea via enzymatic reaction.

8:52AM C31.00005 'Superfluid’ vs collective motion in model active suspensions. ALEXANDER MOROZOV, VIKTOR ŠKULTETY, School of Physics & Astronomy, University of Edinburgh, UK — Systems comprising active particles often exhibit non-zero steady-state currents, and possess unique mechanical and transport properties that are absent from their passive counterparts. One of the most striking examples of such properties is the recent experimental observation of a vanishingly small shear viscosity in suspensions of swimming bacteria. Here, we present our recent results on the connection between this phenomenon and the onset of collective motion in bacterial suspensions. We find that confinement strongly influences both phenomena, and that the apparent viscosity drops to zero before the transition to collective motion. We compare our predictions against the active gel theory, and discuss their relevance to recent experiments.
Two separate mathematical models are proposed to study these behaviors. The formation of microtubule network and its uniform contraction. The second phase is dominated by the steady state flow established afterwards.

In this talk, we use an optically-controlled active matter system, consisting of stabilized microtubule filaments and kinesin motors, to demonstrate a series of systems of biological or synthetic molecules are capable of spontaneously organizing into structures and generating global flows while lacking segregation, movement) by organizing the activity of force-generating, active molecules in time and space. Most experimental active matter systems of biological or synthetic molecules are capable of spontaneously organizing into structures and generating global flows while lacking the spatiotemporal control found in cells, limiting their utility for studying non-equilibrium phenomena and bioinspired engineering. Here, we show how to use this method to simulate deformable droplets of active fluids. When multiple drops are immersed in a Stokesian fluid, their melting is a complex phenomenon. Here we investigate the formation and melting of a nonequilibrium crystal, an active vortex crystal, in a minimum continuum model for active matter. Using simulations, we find spontaneously emerging vortex crystal solutions after an extended turbulent transient. These active vortex crystals melt into a turbulent active fluid as the activity or the advection, the two free parameters in the model, are changed. We map out the phase diagram and systematically characterize the melting transition as a function of both the parameters. Depending on the path through the parameter space, the melting exhibits diverse transition scenarios; it may proceed through a hysteretic marginal stability region or an intermediate hexatic phase. Our results indicate that crystalline phases in active matter share similarities with their equilibrium counterparts.

Two-dimensional crystals show several intriguing properties. For instance, crystals in equilibrium systems lack long-range positional order and their melting is a complex phenomenon. Here we investigate the formation and melting of a nonequilibrium crystal, an active vortex crystal, in a minimum continuum model for active matter. Using simulations, we find spontaneously emerging vortex crystal solutions after an extended turbulent transient. These active vortex crystals melt into a turbulent active fluid as the activity or the advection, the two free parameters in the model, are changed. We map out the phase diagram and systematically characterize the melting transition as a function of both the parameters. Depending on the path through the parameter space, the melting exhibits diverse transition scenarios; it may proceed through a hysteretic marginal stability region or an intermediate hexatic phase. Our results indicate that crystalline phases in active matter share similarities with their equilibrium counterparts.

9:31AM C31.00008 Experimental characterization and modeling of contractile behavior and fluid flows in an optically-controlled microtubule network

9:44AM C31.00009 Engineering microfluidic flow networks and probing self-organisation with light-controlled active suspensions

9:57AM C31.00010 Active yielding in extensile active gels

8:00AM C32.00001 High-accuracy simulations of thousands of deformable, interacting, active droplets
Speaks to swimming and crawling near surfaces with rough boundaries. To the direction of undulatory waves on the body, and body rotation, and depends critically upon the distance to the wall(s). The theory also

SPAGNOLIE, University of Wisconsin-Madison — Surfaces with anisotropic mobilities present an intriguing opportunity for passively influencing driven flow. For the vortex flow, we investigate the effects of SwIMs that encircle the vortex centers. We also test predictions of chaotic trajectories of active tracers in the flow. For the hyperbolic flows, we investigate the structure of the barriers as a function of the imposed flow magnitude. Vortex chain flows. We test theories that predict “swimming invariant manifolds” (SwIMs) that act as one-way barriers that impede the motion of active tracers in the flow. In the hyperbolic flow, we investigate the structure of the barriers as a function of the imposed flow magnitude. For the vortex array, we find both regular and chaotic trajectories of swimmers, including trajectories that exhibit long-range transport. This stands in stark contrast to passive tracers in the same flow, which move on localized, regular orbits when the flow is time-independent. The SwIMs are shown to form leaky one-way barriers to swimmers, and we identify additional phase-space structures—namely, periodic orbits and invariant tori—which regulate the chaotic transport of swimmers throughout the vortex array.

Merced, TOM SOLOMON, Bucknell University — We present experiments on the effects of imposed, laminar fluid flows on the motion of active (self-propelled) tracers. The active tracers are bacillus subtilis bacteria, including a wild-type strain and two variations, one with the GFP mutation and one with a smooth-swimming mutation for which the microbe doesn’t tumble. The imposed flows are simple hyperbolic flows and vortex chain flows. We test theories that predict “swimming invariant manifolds” (SwIMs) that act as one-way barriers that impede the motion of active tracers in the flow. For the hyperbolic flows, we investigate the structure of the barriers as a function of the imposed flow magnitude. For the vortex flow, we investigate the effects of SwIMs that encircle the vortex centers. We also test predictions of chaotic trajectories of smooth-swimming tracers for time-independent, two-dimensional flows.

Supported by NSF Grant CMMI-1825379.

8:26AM C32.00003 Towards the rheology of a concentrated array of spherical squirrmers, TIM PEDLEY, University of Cambridge, TAKUJI ISHIKAWA, Tohoku University, DOUGLAS BRUMLEY, University of Cambridge — Continuum models of dilute suspensions of swimming micro-organisms are well established and can incorporate external (gravitational) forces and torques as well as the particle stress generated by the swimming activity [1]. In a semi-dilute suspension of steady spherical squirrmers, hydrodynamic and steric interactions between cells can be computed in a pairwise manner, and Stokesian Dynamics (SD) has been developed for higher concentrations. The stress response to externally applied simple shear has been computed for semi-dilute suspensions [2]. Recently we have examined the stability of a concentrated planar array of identical bottom-heavy squirrmers, accounting for cell-cell interactions by the use of lubrication theory [3]. Here we seek to extend this theory to externally driven simple shear, in order to represent the macroscopic shear stress and normal stresses as functions of the shear-rate, the orientation of the applied shear to gravity, and the dimensionless parameters of the squirrmers in motion. Preliminary results are compared with those of a full SD computation. References: [1] Pedley, T.J. & Kessler, J.O. (1992) Ann. Rev. Fluid Mech. 24:313-358. [2] Ishikawa, T. & Pedley, T.J. (2007) J.Fluid Mech. 588:399-435. [3] Brumley, D.R. & Pedley, T.J. (2019) Phys.Rev.Fluids 4:053102

9:05AM C32.00006 Swimming near Tensorial Slip Surfaces, CHRISTOPHER DUPRE, SAVERIO SPAGNOLIE, University of Wisconsin-Madison — Surfaces with anisotropic mobilities present an intriguing opportunity for passively influencing the motion of microorganisms or active particles in their presence. We study the self-propulsion of a classical model of microorganism locomotion, Taylor’s swimming sheet, in the presence of one or two tensorial-slip surfaces. The swimming speed generally includes a lateral motion relative to the direction of undulatory waves on the body, and body rotation, and depends critically upon the distance to the wall(s). The theory also speaks to swimming and crawling near surfaces with rough boundaries.

9:18AM C32.00007 Upstream swimming of active Brownian particles in pressure-driven flow, ZHIWEI PENG, JOHN BRADY, California Institute of Technology — Active Brownian particles (ABPs) accumulate at boundaries that confine them owing to their persistent self-propulsion. In the presence of a pressure-driven flow, the fluid vorticity tends to rotate ABPs toward the upstream direction. As a result, ABPs in pressure-driven flow can exhibit a net upstream motion. In this work, we use continuum theory and Brownian dynamics simulation to study the effects of pressure-driven flow on the dynamics of ABPs. In particular, we quantify the net mean speed of ABPs in a channel, which results from a competition between downstream fluid advection and upstream swimming. We characterize the transition between net upstream and downstream motion as a function of the flow speed, Brownian diffusion and the intrinsic swimming speed of ABPs. Our results show that the interplay between self-propulsion, fluid vorticity and confinement provides a robust mechanism for upstream motility.

This work is supported by NSF Grant CBET 1803662.
and unviscid flow, and intermittent viscosity, GIUSEPPE NEGRO, LIVIO NICOLA CARENZA, Dipartimento di Fisica, Università degli Studi di Bari and INFN, Sezione di Bari, via Amendola 173, Bari, I-70126, Italy, ANTONIO LAMURA, Istituto Applicazioni Calcolo, CNR, Via Amendola 122/D, I-70126 Bari, Italy, GIUSEPPE CONNELLA, Dipartimento di Fisica, Università degli Studi di Bari and INFN, Sezione di Bari, via Amendola 173, Bari, I-70126, Italy., ADRIANO TIRIBOCCHI, Center for Life Nano ScienceLa Sapienza, Istituto Italiano di Tecnologia, 00161 Rome, Italy — Active fluids are systems where active components present in the fluid (microtubules with molecular motors such as kinesin or actomyosin bundles) display interesting collective ordering properties. Active fluids also exhibit peculiar rheological properties. Depending on the characteristic of the active stress, activity is capable to heighten viscosity, enough to develop shear-thickening properties in contractile systems or induce in extensile suspensions a superfluid regime under suitable condition. We study, by lattice Boltzmann methods, the rheological behavior of an emulsion made of an active polar component and an isotropic passive fluid. Different flow regimes are found by varying the values of shear rate and extensile activity (occurring, e.g., in microtubule-motor suspensions). By increasing activity, a first transition occurs from linear flow regime to spontaneous persistent unidirectional macro-scale flow, followed by another transition either to (low shear) intermittent flow regime with coexistence of states with positive, negative, and vanishing apparent viscosity, or to (high shear) symmetric shear thinning regime.

Generalized rheotaxis of active particles in confined Stokes flow, WILLIAM USPAL, University of Hawai'i at Manoa, Dept. of Mechanical Engineering — We consider spherical active particles exposed to shear flow near a planar surface. The swimming activity of a particle gives rise to interactions (e.g., hydrodynamic or chemical interactions) with the surface that couple back to the motion of the particle. Via a dynamical systems approach, we show that a robust directional response can emerge from the interplay of external flow and near-surface swimming activity. For instance, depending on the external flow strength and the character of the swimming activity of the particle, the particle can align against the flow direction (upstream rheotaxis) or nearly perpendicular to it (cross-stream rheotaxis). As an instructive and analytically tractable example, we apply our findings to the squirmer model of a mechanically actuated microswimmer. Using far-field approximations, lubrication theory, and exact numerical calculations, we identify the conditions on the squirming mode amplitudes to obtain the various steady states. Finally, we discuss collective rheotaxis of a group of squirmers. Overall, our findings demonstrate that microswimmers can exhibit surprisingly rich behavior when operating in confined flows, which occur in many of their envisioned applications.

Spatiotemporal Control of an Extensile Active Nematic Suspension1, MICHAEL NORTON, MICHAEL HAGAN, SETH FRADEN, Brandeis University — Active nematic suspensions are self-driven fluids that exhibit rich spatiotemporal dynamics characterized by director field buckling, defect nucleation/annihilation and chaotic trajectories of those defects. Towards developing experimental methods for controlling these dynamics, we consider an optimal control problem which seeks to find the spatiotemporal pattern of active stress strength required to drive the system towards a desired director field configuration. As an exemplar, we consider an extensile active nematic fluid confined to a disk which, in the absence of control, produces two topological defects that perpetually circulate, and seek the time-varying active stress field that drives the system to circulate in the opposite direction.

Feedback control of Marangoni instability in a thin film1, ANNA SAMOLOVA, ALEXANDER NEMPOMYASHCHY, Technion - Israel Institute of Technology — We use the feedback control for the suppression of Marangoni instability in a thin film heated from below. Our keen interest is focused on the oscillatory mode that was recently revealed in the case of the thermally insulated substrate. We apply the lubrication approximation to derive nonlinear amplitude equations which govern the coupled evolution of the layer thickness and the characteristic temperature. To stabilize the no-motion state of the film we apply the linear feedback control. The principle of such control is that we feed the information about instability back into the dynamics by changing the heat flux through the substrate that can affect the instability. Linear analysis shows that the control of the temperature deviation at the deformable free surface stabilizes the thin film heated from below effectively. To render the subcritical instability supercritical we apply the quadratic feedback control. The weakly nonlinear analysis of the amplitude equations is provided for two types of regime: a pair of the travelling waves and a pair of the standing waves. The conditions of stable supercritical solutions are obtained for both types of regime.

Linear stability analysis of plane Poiseuille-Couette flow over a permeable surface1, SAMAN HOOSHYAR, PARISA MIRBOD, University of Illinois at Chicago — The instability of plane Poiseuille-Couette flow (PCF) over a porous medium has been studied extensively due to its wide range of applications in environmental engineering, oil and pharmaceutical industries, and geophysical problems. This study investigates the effect of imposing Couette to the instability of plane Poiseuille flow over a porous surface. We performed linear stability analysis to analyze the impact of upper moving wall on the porous media with different geometrical parameters and to obtain the most unstable perturbation modes. The Navier-Stokes and Brinkman equations are coupled to characterize the behavior of the fluid and porous layers, respectively. It was found that the presence of Couette flow always destabilizes the Poiseuille flow when the velocity of the upper moving wall is smaller than the cutoff velocity, defined as the velocity at which the flow stabilizes and it is a function of the depth ratio and porous resistivity. The flow becomes more stable as the wall velocity exceeds the cutoff velocity. Also, increasing the wall velocity results in an instability mode shift from the fluid layer mode to the porous layer mode.

Feedback control of Marangoni instability in a thin film1. This research was supported by the Israel Science Foundation (grant No. 843/18)

Linear stability analysis of plane Poiseuille-Couette flow over a permeable surface1. This research was supported by the Army Research Office (ARO) under Award No. W911NF-18-1-0356

---

1We acknowledge support from NSF DMREF-1534890 and NSF MRSEC, Bioinspired Soft Materials, DMR-1420382.

1This research was supported by the Army Research Office (ARO) under Award No. W911NF-18-1-0356.
in the case of a pre-coated glass, the famous “wine tears” emerge from a reverse undercompressive shock originating at the meniscus. We observe both compressive and undercompressive waves in new experiments and we argue that, and the behavior of the meniscus. In the lubrication limit we obtain an equation that is already well-known for rising films in the presence of inertia. The nonlocal effect of the electric field solution in particular is examined in detail.

Via comparison with Direct Numerical Simulations, we show that good accuracy can be maintained even for thick films at moderate levels of inertia. The nonlocal effect of the electric field solution in particular is examined in detail.


8:39AM C33.00004 Electrostatic control of instabilities in thick liquid layers, ALEXANDER WRAY, University of Strathclyde, RADU CIMPEANU, University of Oxford — We apply a recently developed method for modelling thick flows (Wray et al., 2017) to the Moffatt problem for flow on the outside of a rotating cylinder, and the control thereof using electric fields. Via comparison with Direct Numerical Simulations, we show that good accuracy can be maintained even for thick films at moderate levels of inertia. The nonlocal effect of the electric field solution in particular is examined in detail.

8:52AM C33.00005 Low-order modelling of non-axisymmetric flow on the exterior of a fibre1, JAMES REILLY, ALEXANDER WRAY, STEPHEN WILSON, Strathclyde University, OMAR MATAR, Imperial College London — Flows on the outside of fibres have received extensive attention over the years. Even in the Stokes limit they exhibit interesting behaviour due to dual role of capillarity, exhibiting simultaneously both a destabilising “hoop stress” in the azimuthal direction as well as a more conventional stabilising effect in the axial direction. In this limit, the flow is always axisymmetric. However, for sufficiently thick films and with a sufficiently high Reynolds number, non-axisymmetric instabilities are observed. While this has been predicted using linear stability theory and confirmed using direct numerical simulations and experimentation, it has proven problematic to model. Existing models have either required that films be both thin and close to threshold (Shiang and Sivashinsky, 1982) or that the flow be axisymmetric (Ruyer-Quil et al., 2008). Using the methodology of Wray et al., 2017, we develop a fully nonlinear model for thick flow down the outside of a fibre at moderate levels of inertia. We compare its predictions against those from numerical computations in both the linear and nonlinear regimes to assess its validity, before using it to elucidate the complex interplay of viscosity, capillary, inertia, and gravity that gives rise to the observed flow patterns.

1Royal Academy of Engineering, UK, Research Chair for OKM

9:05AM C33.00006 Nonlinear evolution of a dewetting bilayer thin film on a soft-gel layer, PUSHPAVANAM SUBRAMANIAM, DINESH BHAGAVATULA, Indian Institute of Technology Madras — The nonlinear evolution of dewetting bilayer thin film on a soft-gel is investigated in this work. The fluids are considered to be Newtonian and the soft-gel-layer is modelled as a linear viscoelastic solid. The free surface of the top fluid is exposed to air. The van der Waals attractive forces between the soft-gel layer and the fluids are the primary driving forces responsible for breaking of the thin films. Short range repulsive forces play a key role in the dynamics of the system for very thin bilayers. The nonlinear evolution equations for this system are derived using a lubrication approximation. Linear stability analysis of the system confirms two long wave instabilities one at the liquid-liquid interface and other at the free surface arise in the system. The liquid-liquid instability is dominant when the attractive forces between the soft-gel-layer and the bottom fluid are strong. For a range of parameters both instabilities coexist in the system. The short range repulsive forces can suppress both these instabilities. The soft-gel layer has a destabilizing effect on the long wave instabilities.

9:18AM C33.00007 Asymptotics regimes in elastohydrodynamic and stochastic levelling on a viscous film1, CHRISTIAN PEDERSEN, Department of Mathematics, University of Oslo, Oslo, Norway, JOHN NIVEN, Department of Physics and Astronomy, McMaster University, Hamilton, Ontario, Canada, THOMAS SALEZ, University of Bordeaux, CNRS, LOMA, Talence, France, KARI DALNOKI-VERESS, Department of Physics and Astronomy, McMaster University, Hamilton, Ontario, Canada, ANDREAS CARLSON, Department of Mathematics, University of Oslo, Oslo, Norway — We study the relaxation dynamics of an elastic plate resting on a thin viscous film that is supported by a solid substrate. By combining scaling analysis, numerical simulations and experiments, we identify asymptotic regimes for the elastohydrodynamic leveling of a surface perturbation when the flow is driven by either elastic bending of the plate or thermal fluctuations. In both cases we identify two distinct regimes when the perturbation height is either much larger or much smaller than the thickness of the pre-wetted viscous film. Our analysis reveals a distinct crossover between the similarity exponents with the ratio of the perturbation height to the film height.

1The financial support by the Research Council of Norway (project grant 263056), the Natural Science and Engineering Research Council of Canada and the Joliot Chair of ESPCI Paris is gratefully acknowledged.

9:31AM C33.00008 A theory for undercompressive shocks in tears of wine1, HANGJIE JI, YONATHAN DUKLER, CLAUDIA FALCON, ANDREA BERTOZZI, Department of Mathematics, University of California, Los Angeles — We revisit the tears of wine problem for thin films in water-ethanol mixtures and present a new model for the climbing dynamics. The new formulation includes a Marangoni stress balanced by both the normal and tangential components of gravity as well as surface tension which lead to distinctly different behavior. The prior literature did not address the wine tears but rather the behavior of the film at earlier stages and the behavior of the meniscus. In the lubrication limit we obtain an equation that is already well-known for rising films in the presence of thermal gradients. Such models can exhibit non-classical shocks that are undercompressive. We present basic theory that allows one to identify the signature of an undercompressive wave. We observe both compressive and undercompressive waves in new experiments and we argue that, in the case of a pre-coated glass, the famous “wine tears” emerge from a reverse undercompressive shock originating at the meniscus.

1Simons Foundation Math+X investigator award number 510776, YD was supported by National Science Foundation Graduate Research Fellowship under Grant No. DGE-1650004
9:44AM C33.00009 Dewetting through Forests of Micropillars. YU QIU, KE XU, Massachusetts Institute of Technology, AMIR PAHLAVAN, Princeton University, RUBEN JUANES, Massachusetts Institute of Technology — When a low-viscosity nonwetting fluid displaces a higher-viscosity wetting liquid in a rough surface consisting of a forest of micropillars, a film of the viscous defending fluid is left behind the advancing fluid-fluid front. Here, we investigate experimentally the dewetting of this residual film, and we contrast it with the dewetting that occurs on a smooth surface. We find that there are three regimes: “no film” at low capillary number; “thick film” at sufficiently high capillary number, and a novel “thin film” regime at intermediate capillary number. In this latter regime, the dewetting occurs as the residual viscous fluid retracts as a film of uniform thickness through (and not over) the micropillars. We propose necessary geometric conditions for the emergence of this new dewetting dynamics, and delineate the different dewetting regimes on a phase diagram, which quantitatively matches lab experiments conducted on a microstructured Hele-Shaw cells. The long-term state of the system induced by dewetting is a pattern of residual film clusters confined in the micropillar forest—a fluid arrangement that impacts macroscopic hydrodynamics and heat transfer efficiency.

9:57AM C33.00010 Deformation and dewetting of liquid films under gas jets. CHINASA OJIAKO, Loughborough University, RADU CIMPEANU, Oxford University, HEMÁKA BANDULASENA, ROGER SMITH, DMITRI TSHELUIKO, Loughborough University — We study the deformation and dewetting of a liquid film in a cylindrical beaker under the influence of an impinging gas jet. To obtain initial insight into relevant regimes and timescales of the system, we first derive a reduced-order model (a thin-film equation) based on the long-wave assumption and on appropriate decoupling the gas problem from that for the liquid [1-3] and taking into account a disjoining pressure [4]. We also perform direct numerical simulations (DNS) of the full governing equations using two different approaches, the Computational Fluid Dynamics (CFD) package in COMSOL and the volume-of-fluid Gerris package. The DNS are used to validate the results for the thin-film equation and also to investigate the regimes that are beyond the range of validity of this equation. We additionally compare the computational results with experiments and find good agreement.

Sunday, November 24, 2019 8:00AM - 10:10AM – Session C34 Flow Instability: Rayleigh-Taylor | 616 - Irmgard Bischofberger, MIT

8:00AM C34.00001 Pattern formation of a thin film flowing under an inclined plane1. PIER GIUSEPPE LEDDA, GAETAN LERISSON, GIOELE BALESTRA, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities, Ecole Polytechnique Federale de Lausanne, CH-1015 Lausanne, Switzerland — We discuss the pattern formation of a thin film flowing under an inclined planar substrate, combine theoretical, experimental and numerical results. The phenomenon is related to the Rayleigh-Taylor instability, in which one heavier fluid is placed above a lighter one. When an upper wall and the substrate inclination are considered, a variety of patterns are observed. The natural and forced dynamics of the flat film to spanwise perturbations and the resulting non-linear structures are studied; in both cases, spanwise-periodic, streamwise-aligned structures, called rivulets, arise. The impulse response of a flat film is numerically and experimentally studied. We analyze the linear response, which does not show any preferential direction; a weakly non-linear model highlights however the selection of the streamwise structures. The fully non-linear evolution leads to a steady pattern characterized by fully saturated rivulets, the profile of which is analyzed in detail. A secondary stability analysis reveals the presence of a range of parameters in which only rivulets are observed, in agreement with the experimental observations. Outside of this range, lenses appear on the rivulets, which may eventually drip.

1Swiss National Science Foundation (Grant No. 200021_178971)

8:13AM C34.00002 A Buoyancy–Shear–Drag-Based Turbulence Model for Rayleigh–Taylor, Reshocked Richtmyer–Meshkov, and Kelvin–Helmholtz Mixing1. OLEG SCHILLING, Lawrence Livermore National Laboratory — A phenomenological turbulence model for Rayleigh–Taylor, reshocked Richtmyer–Meshkov, and Kelvin–Helmholtz instability-induced mixing is developed using a general buoyancy/shear/drag model. Analytical solutions are used to calibrate the coefficients to predict specific values of the mixing growth parameters and exponents. The model is applied to the three instability cases to demonstrate its utility. It is shown that the numerical solutions of the model calibrated using specific values of the instability growth parameters and exponents: (1) produces mixing layer widths in agreement with the expected self-similar growth power laws; (2) gives spatiotemporal profiles of turbulent fields that are expected and consistent with previous results; (3) predicts the expected power-law growths and decays of the spatially-integrated turbulent fields, and; (4) gives spatiotemporal profiles of the mean fields that are expected and consistent with previous results.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

8:26AM C34.00003 Combined effects of finite plate thickness and acceleration rates on Rayleigh-Taylor instability in elastic-plastic materials. ARINDAM BANERJEE, RINOSH POLAVARAPU, AREN BOYACI, Lehigh University — Majority of theoretical and computational studies of Rayleigh-Taylor instability (RTI) in solid-fluid interfaces assume the materials to be incompressible with infinite/semi-infinite thickness subjected to a constant driving acceleration. A recent theoretical study by Piriz et al. (PRE textbf{95}, 053108, 2017) has explored finite thickness effects using an impulsive acceleration profile where the initial rate of increase in the driving pressure is nearly infinite. In previous studies, our group has addressed the independent effects of the finite plate/slab thickness and finite rate of increase in driving acceleration (pressure). In this talk, we will address the combined effects of both the aforementioned parameters on RTI by employing the soft solid (mayonnaise)-air interface using our rotating wheel experimental setup. A set of experiments was run at four different acceleration rates in combination with three test-section container halves of varying depths using initial conditions with different wavelengths and amplitudes. The aggregate effects of these parameters on interface growth were quantified in terms of instability acceleration, which signifies the material transition from elastic-plastic regime to the viscous regime. In addition, the growth rates for each experiment are determined and compared to the existing theoretical models which tackle the RTI problem in solids.

1This work is funded through a DOE-SSAP Grant #DE-NA0003195 and a LANL Subcontract #370333.
8:39AM C34.00004 Suppression of the Rayleigh-Taylor instability in a confined geometry

We study the Rayleigh-Taylor instability of two miscible fluids in a Hele-Shaw geometry; confined in a thin gap, of size \( b \), between two large flat plates. Using this geometry, we inject a fluid into another with a different density as to produce an unstable situation in which a heavy fluid initially resides above a layer of lighter fluid. Below a critical gap spacing, \( b_c \), we find that no Rayleigh-Taylor fingers form despite the fluid density gradient that typically instigates the instability. We use simulations, validated by comparison with experiments, to determine \( b_c \) as a function of the difference in densities \( \Delta \rho \), the viscosities \( \eta \), and diffusivities \( D \). We argue that this critical confinement scale is set by a competition between destabilizing buoyancy forces and stabilizing effects of viscosity and diffusion. An argument based on dimensional analysis gives scaling exponents consistent with the observed results, \( b_c \sim (D\eta/g\Delta \rho)^{1/3} \). In addition to the critical gap, we measure the characteristic wavelength and onset time in this confined geometry and compare it to the theoretical predictions for the Rayleigh-Taylor instability in open space.

8:52AM C34.00005 Group theory analysis of early-time scale-dependent dynamics of the Rayleigh-Taylor instability with time varying acceleration

We consider the long-standing problem of Rayleigh-Taylor instability with variable acceleration, and focus on the early-time scale-dependent dynamics of an interface separating incompressible ideal fluids of different densities subject to a power-law function of time for a spatially extended three-dimensional flow periodic in the plane normal to the acceleration. By employing group theory analysis, we discover two distinct sub-regimes of the early-time dynamics depending on the exponent of the acceleration power-law. The time scale and the early-time dynamics are set by the acceleration for exponents greater than \(-2\), and by the initial growth-rate (due to, e.g., initial conditions) for exponents smaller than \(-2\). At the exponent value \(-2\) a transition occurs from one sub-regime to the other with varying acceleration strength. For a broadband range of acceleration parameters, the instability growth rate is explicitly found, the dependence of the dynamics on the initial conditions is investigated, and theory benchmarks are elaborated.

The authors thank the National Science Foundation (USA) and the University of Western Australia (AUS)

9:05AM C34.00006 Rayleigh-Taylor Instability Between Unequally Stratified Layers

We consider the classical Rayleigh-Taylor instability between two otherwise stably stratified layers that a new growth is limited by the stratification and can be arrested before the instability reaches the boundaries. Here we study the intermediate case where one of the two layers is stably stratified and the other homogeneous, thus introducing an asymmetry where the instability grows into one layer without restriction, but is eventually arrested by the stratification in the other layer. Experimental measurements of the density and velocity fields show both similarities and differences compared with both the limiting cases in terms of both the evolution of the height of the mixing zone and the final stratification achieved.

9:18AM C34.00007 Optimal perturbations for linear stability of two fluid columns of different densities subject to gravity

We study the linear stability of a vertical interface separating two fluid columns of different densities under the influence of gravity. Initially, we assume quasi-steady state (QSSA) of the base flow and pose the problem as an eigenvalue problem. This result in an initial condition that leads to the maximum growth of disturbances at a finite time. Preliminary results indicate that the perturbation energy of wave modes with small wave numbers may experience substantial transient growth prior to decaying asymptotically in time, despite the assumption of the linearized problem. It is also found that the maximum growth is achieved for an order of magnitude higher than that of the non-optimized case. The sensitivity of perturbation growth with respect to initial time, density, and viscosity ratios will be investigated.

9:31AM C34.00008 Experimental investigation of Rayleigh—Taylor mixing in gases using simultaneous PIV-PLIF

Particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) — are employed to obtain mixing width and simultaneous velocity-density data. PLIF using acetone is implemented for the first time for convective-type (flowing) statistically stationary RT experiments with gases. Velocity and density statistics, and their correlations (\( u', v', \rho', \rho' v' \)) are presented. As Atwood number for current experiments exceeds the widely accepted \( A = 0.1 \) limit for Boussinesq approximation, non-Boussinesq effects and anisotropy effects at this Atwood number are evaluated using metrics like higher-order moments (skewness, kurtosis) and anisotropy tensor. Results from current experiments are compared with existing turbulent RT mixing models (like BHR models). Reference: AKULA, B. & RANJAN, D. 2016 Dynamics of buoyancy-driven flows at moderately high Atwood numbers. Journal of Fluid Mechanics 795, 313–355.

9:44AM C34.00009 The adjoint Rayleigh criterion in compressible reacting flow instabilities

Thermoacoustic oscillations are one of the most challenging flow instabilities faced by the gas turbine and rocket motor industry. The instability mechanism is described in the time domain by the Rayleigh criterion. In this contribution, the Rayleigh criterion is interpreted in the frequency domain by deriving functional formulae for the eigenvalue. The first variation of the Rayleigh criterion is calculated both in the time and frequency domain, both with and without Lagrange multipliers (adjoint variables). The Lagrange multipliers are physically interpreted with the system's observables, e.g., pressure, velocity, temperature. Finally, the adjoint Rayleigh criterion is proposed. The relations and criteria proposed can enable the calculation of adjoint sensitivities from measurable quantities in experiments. The methodology proposed is versatile and can be applied to other problems in flow instability that are tackled by adjoint analysis.

Royal Academy of Engineering Research Fellowships Scheme
Is it possible to design a Stokes flow vortex that would trap an active particle? We address mathematically the fate of swimmers modelled as nodes are enough to tilt the swimmers around obstacles. We address this problem with a combination of theory, simulations and experiments. We

ZHANG, MICHAEL WARD, New York University, MICHAEL J. SHELLEY, Flatiron Institute — We revisit the problem of microswimmer interfacial coherent structures with time-dependent acceleration. We agree with existing observations and elaborate new benchmarks for the future.

The authors thank the National Science Foundation (USA) and the University of Western Australia (AUS)

Sunday, November 24, 2019 8:00AM - 10:10AM – Locomotion 617 - Bhargav Rallabandi, University of California, Riverside

8:00AM C35.00001 How to catch a microswimmer. IVAN TANASJIEVIC, ERIC LAUGA, University of Cambridge — An active particle swimming in a flow undergoing solid-body rotation always ends up diverging away from the centre of rotation. Is it possible to design a Stokes flow vortex that would trap an active particle? We address mathematically the fate of swimmers modelled as spheres, or prolate swimmers, swimming with prescribed velocities in a number of elementary vortical flows in 2D and 3D. We predict theoretically the existence of bounded orbits, which are confirmed by numerical simulations. We also address the role of fluctuations for the swimmer. These results open the possibility of designing microswimmer traps, thereby facilitating biophysical studies of cell motility.

8:13AM C35.00002 Self-propulsion of chemically active colloids. BHARGAV RALLABANDI, University of California, Riverside, FAN YANG, HOWARD STONE, Princeton University — Phoretic particles are able to propel in the presence of externally applied or self-generated chemical gradients. Focusing on the non-Brownian limit, we discuss the autonomous motion of colloids that generate chemical gradients through surface reactions and consequently self-propel via diffusiophoresis. We develop a general framework by combining truncated multipole expansions for the hydrodynamic and chemical fields with the Lorentz reciprocal theorem. Applying the framework to particles with uniform surface chemical fluxes, we find that the particles translate primarily through chemical interactions, whereas hydrodynamic interactions are typically subdominant. We discuss the utility of truncated expansions in accurately describing particle trajectories, while also contrasting autophoretic motions in ionic and non-ionic solvents. We apply the model to a mixture of two particle populations and find behaviors that include pair-chasing, attraction and cluster formation.

8:26AM C35.00003 Propulsion of spherical microparticles through spontaneous symmetry breaking in mucus. HENRY FU, University of Utah, LOUIS ROGOWSKI, Southern Methodist University, JAMEL ALI, Florida Agricultural and Mechanical University-Florida State University, XIAO ZHANG, MINJUN KIM, Southern Methodist University — Symmetries have long been used to understand when propulsion is possible in microscale systems. For biological force- and torque-free swimmers the Scallop theorem applies kinematic reversibility of Stokes flow has been to understand which swimming strokes are asymmetric under time-reversal and capable of propulsion. Currently, artificially propelled magnetic micro- and nanoparticles are being utilized in a variety of techniques including hyperthermia, drug delivery, and magnetic resonance imaging. Rotation of rigid magnetic particles by an external magnetic field are a promising category of such artificial propulsion. So far it has been expected that spherical beads are too symmetric to be propelled in this fashion, but here we present experiments demonstrating that rotating spherical magnetic beads in a solution of mucin seem to display a spontaneous symmetry-breaking propulsion. We perform a perturbative analysis of a rotating sphere in a nonlinear polymeric fluid that elucidates the physical mechanism behind such a symmetry breaking.

8:39AM C35.00004 Locomotion of a rotating cylinder pair with periodic gaits at low Reynolds numbers. LINGBO JI, WIM M. VAN REES, Massachusetts Institute of Technology — We investigate the effect of periodic gaits on the self-propulsion of two side-by-side rotating cylinders at low Reynolds numbers. This cylinder-pair model serves as a prototype for engineered micro-swimmers due to its simple design and operation. To study periodic gaits, we impose a zero-net-rotation constraint on each cylinder for each swimming cycle. By numerically solving the Stokes flow around the cylinders, we can identify the optimal rotation patterns that enable the cylinder pair to travel the largest distance for a range of energy budgets. We extend this study to three dimensions by investigating the optimal swimming strategies for pairs of spheroids within a range of aspect ratios, and compare their performance to traditional Purcell swimmers and other locomotion strategies at low Reynolds numbers.

8:52AM C35.00005 Microswimmer and obstacle interactions mediated by pressure fields. FLORENCIO BALBOA USABIAGA, Flatiron Institute, QUENTIN BROSSEAU, Courant Institute, New York University, ENKELEIDA LUSHI, New Jersey Institute of Technology, YANG WU, New York University, LEIF RISTROPH, Courant Institute, New York University, JUN ZHANG, MICHAEL WARD, New York University, MICHAEL J. SHELLEY, Flatiron Institute — We revisit the problem of microswimmer orientation near boundaries. We will show how elongated swimmers create high and low pressure nodes around them and how these pressure nodes are enough to tilt the swimmers around obstacles. We address this problem with a combination of theory, simulations and experiments. We use the Rigid multiblob method to model phoretic swimmers near walls and compare the predictions with experiments of bimetallic micro-rods swimming in hydrogen peroxide solutions. Finally, we will use our mechanical explanation to suggest how phoretic swimmers can be designed to have preferred hydrodynamic interactions with walls.

This work was supported by NSF-MRSEC program DMR-1420073, and NSF Grants DMS-1463962 and DMS-1620331.
9:05AM C35.00006 Hydrodynamics of Microswimmers Trapped at Fluid-Fluid Interfaces, NICHOLAS CHISHOLM, KATHLEEN STEBE, University of Pennsylvania — Microswimmers on or near fluid interfaces have pronounced changes in their motion, in their interactions with neighboring colloids, and in their collective behaviors. These motions and interactions can be harnessed to direct colloidal assembly or alter interfacial transport conditions. However, the manner in which a fluid interface alters microswimmer motion is not yet understood. To address this gap, we develop a flow singularity model to identify leading order modes for active colloids on interfaces in the limit of small Reynolds and capillary numbers. We discuss a “clean” interface in which there is no jump in tangential stresses at the interface. We also consider an incompressible interface, as is typical for colloids on interfaces in the presence of surfactant. Thereafter, we examine interactions between a microswimmer and passive “tracer” particles for two microswimmer trajectory types: swimming in a straight line or in circular paths. These results will be useful in future work on the use of active colloids to direct and enhance transport at interfaces.

9:18AM C35.00007 Active Particles Powered by Quincke Rotation in a Bulk Fluid, DEBASHIS DAS, ERIC LAUGA, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — How are groups of living organisms such as flocks of birds, sheep, schools of fish and bacterial colonies able to self-organize and display collective motion? This question has fascinated scientists for decades and has given rise to the new field of ‘active matter’. One of the key features of active matter is that it is composed of self-propelled units that move by consuming energy from their surrounding with a direction of self-propulsion typically set by their own anisotropy (shape or function) rather than by an external field. In this work, we show how an electrohydrodynamic instability called Quincke rotation can be exploited to develop an ideal self-propelled particle in a bulk fluid. Dielectric particles suspended in a weakly conducting fluid are known to spontaneously start rotating under the action of a sufficiently strong uniform DC electric field due to Quincke rotation. This rotation can be converted into translation when the particles are placed near a surface providing useful model systems for active matter. Using a combination of numerical simulations and theoretical models, we demonstrate that it is possible to convert the spontaneous Quincke rotation into spontaneous translation even in the absence of surfaces by relying on geometrical asymmetry instead.

1 European Research Council under the European Unions Horizon 2020 research and innovation programme

9:31AM C35.00008 Harnessing low Reynolds number flow for net migration: Locomotion of a deformable microcapsule by random fluid forces, TAKUJI ISHIKAWA, TAKERU MORITA, TOSHIHiro OMORI,YOHEI NAKAYAMA,SHOICHI TOYABE,Tohoku University — Random noise in low Reynolds number flow has rarely been used to obtain net migration of microscale objects. In this paper, we show that net migration of a microscale object can be extracted from random directionnal fluid forces, by introducing deformability and inhomogeneous density into the object. As a model system, we considered an elastic microcapsule containing fluid and a rigid sphere with different densities. The numerical results showed that the microcapsule could migrate vertically downward when random forces were applied using a migration mechanism based on non-reciprocal body deformation in Stokes flow. We also developed a mathematical framework to describe the deformation-induced migration caused by noise. The proposed theory showed good agreement with the simulation results, and illustrated that drag asymmetry acts like a ratchet to generate net downward motion under noise. These results provide a basis for understanding the noise-induced migration of a micro-swimmer and are useful for harnessing energy from low Reynolds number flow.

1 JSPS KAKENHI Grant No. 17H00853

9:44AM C35.00009 Generalised geometric swimming for Stokes flow, LYndon KOENS, Macquarie University, ERIC LAUGA, University of Cambridge — Shapere and Wilczek first demonstrated that the displacement of a microscopic swimmer was related to path integrals over a gauge field. This field is a function of the swimmer’s configuration and laboratory frame position. For simple 1D swimmers, Stokes theorem can be used to relate the net displacement from a stroke to the area within a curve. This provides an effective method to determine the optimal strokes for displacement. However, a similar visualization is difficult for many swimmers because of complex configuration spaces or non-commuting variables. In this talk I will use a Purcell swimmer to demonstrate how to overcome these issues. These techniques reveal general properties about the displacement of microswimmers while offering a new method to optimise them.

9:57AM C35.00010 Magnetocapillary Swimmers: a Self-Assembled System to Study Locomotion, Transport Cargoes and Mix Fluids, GALIEN GROSJEAN, YLONA COLLARD, University of Liege (ULiege), MAXIME HUBERT, ULiège / Friedrich-Alexander University (FAU), ALEXANDER SUKHOV, Helmholtz Institute Erlangen-Nuremberg (HI-ERN), JENS HARTING, HI-ERN / Eindhoven University of Technology, ANA-SUNCANA SMITH, FAU / Institute Ruder Boskovic, NICOLAS VANDEWALLE, ULiège — Because of capillary forces, small objects floating on a liquid tend to aggregate. Combined with the induced rotation from a magnetic field, this effect can be used to assemble soft-ferromagnetic spheres into tunable structures. When they are exposed to oscillating magnetic fields, these assemblies spontaneously move along the interface. This is due to a breaking of time-reversal symmetry in their adopted shapes. These structures are conceptually simple, chemically inert, and spontaneously form without direct intervention or complex microfabrication process. Therefore, they offer a very wide range of possibilities, from the experimental study of the basic physical principles of locomotion, to the development of complex tasks such as cargo transport and fluid mixing.

1 FNRS PDR grant T.0129.18 / DFG Priority Programme SPP 1726 / FNRS FRIA / FNRS Grant 2.5020.11

Sunday, November 24, 2019 8:00AM - 10:10AM
Session C36 Electokinetic Flows: Induced Charge Flows and Instabilities 618 - Igor Novoselov, University of Washington

8:13AM C36.00002 Asymmetric Rectified Electric Fields Generate Flows that can Dominate Induced-Charge Electrokinetics, S. M. H. HASHEMI AMREI, GREGORY MILLER, WILLIAM RISTENPART, University of California Davis — We derive a generalized induced-charge electrokinetic (ICEK) velocity around a conducting object placed in an arbitrary multimodal electric field. The generalized model allows consideration of asymmetric rectified electric fields (AREFs), which have recently been established to occur in liquids where the ions present have unequal mobilities. Including the AREF yields fluid velocities in which both the direction and the magnitude depend sensitively on the applied potential, frequency, ionic type and strength, and even the exact placement of the object between parallel electrodes. The results provide a new explanation for the long-standing question of flow reversals observed in ICEK systems.

8:26AM C36.00003 Electric-Field Induced Pattern Formation in Layers of DNA Molecules at the Interface between Two Immiscible Liquids1, JOHANNES HARTMANN, STEFFEN HARDT, SICHENG ZHAO, Institute for Nano- and Microfluidics, Technische Universität Darmstadt, ADITYA BANDOPADHYAY, Department of Mechanical Engineering, India Institute of Technology Kharagpur — Electrohydrodynamic/electrokinetic flows play a key role as driving mechanism of pattern formation in colloidal dispersions and at liquid-liquid and liquid-solid interfaces with adsorbed particles. We report an analogous phenomenon of electric field-induced concentration patterns of DNA molecules at the interface of an aqueous two-phase system. The electric field, applied normal to the liquid-liquid interface, drives the molecules to the interface where they get trapped. Hydrodynamic interactions between the molecules arise by electroosmotic flow due to the Debye layer around the polyelectrolytes, leading to pattern formation in the DNA concentration field at the interface. We describe the time evolution of the concentration field by a non-linear integro-differential equation. A linear stability analysis of the equation yields a critical time after which the system destabilizes if exited by a mode of given wavelength. We find that the scaling behavior predicted by theory agrees with experimental results. The presented scheme could be used as an efficient method to pre-concentrate DNA molecules at an interface.

1We wish to acknowledge the support of this work by the German Research Foundation, grants HA2696/37-1 and HA2696/43-2.

8:39AM C36.00004 Enhanced Ion Transport by Controlling Electroconvective on Ion Exchange Membranes with Patterned Structures1, JOONHYEON KIM, SANGHA KIM, RHOKYUN KWAK, Hanyang University, Korea — Patterned structure in fluid systems has been used to generate vortices, enhancing mass transfer. Here, we investigate new role of this vortex promoter, i.e. controlling an electrically driven hydrodynamic instability (i.e., a.k.a electroconvection (EC)) on ion exchange membranes. The patterned structures not only generates vortices as a vortex promoter, but also acts as a shelter to keeping EC from being suppressed by shear flow. To verify these effects, we visualized EC over six different patterns under various applied voltages. The strength of EC was then quantified by visualizing velocity and vorticity fields. In current-voltage response, we found that i) conductive ion flux is inversely proportional to the occupied area of the pattern in Ohmic regime, ii) stronger vortex promotion with a larger vortex area induces a shorter or even negligible length of limiting regime, and iii) the area of sheltered EC benefiting from the pattern is directly proportional to the convective ion flux in overlimiting regime. Considering relationship between ion flux and roles of structures synthetically, isosceles triangle pattern shows the highest ion flux through the membrane as it is the best option for a conductor and the second best for the vortex promoter and EC shelter.

1This work was supported by the Climate Change Response Technology Development Project (NRF-2017M1A2A2047475) and the Basic Research Project (NRF-2019R1C1C1008262) from the National Research Foundation of Korea.

8:52AM C36.00005 Pattern formation of three-dimensional electroconvection on a charge selective surface1, SOOHYEON KANG, RHOKYUN KWAK, Hanyang University, Korea — Electroconvection (EC) has long been known for enhancing ion flux in various electrochemical systems, but its dynamics is yet to be probed in three-dimensions. In this paper, we describe the first laboratory observation of 3-D EC on an exchange membrane and its pattern diversification. Combining experiment and scaling analysis, we successfully categorized three distinguished patterns of 3-D EC according to Reynolds number (Re), electric Rayleigh number (RaE) and Schmidt number (Sc) as i) polygonal, ii) transverse, or iii) longitudinal rolls. If Re increases or RaE decreases, pure longitudinal vortices are presented. On the other hand, transverse rolls are formed between longitudinal rolls, and two rolls are transformed as polygonal rolls at higher RaE or lower Re. In this pattern selection scenario, Sc determines the critical electric Rayleigh number (RaE) for the onset of each transverse or polygonal rolls, resulting RaE∗ ∼ Re∗Sc. We also verify that convective ion flux by EC (represented by an electric Nusselt number, NuE) is fitted to a power law, NuE ∼ RaE0.48Sc−0.39. This work was supported by the Climate Change Response Technology Development Project (NRF-2017M1A2A2047475) and the Basic Research Project (NRF-2019R1C1C1008262) from the National Research Foundation of Korea.

1This work was supported by the Climate Change Response Technology Development Project (NRF-2017M1A2A2047475) and the Basic Research Project (NRF-2019R1C1C1008262) from the National Research Foundation of Korea.

9:05AM C36.00006 Numerical analysis of 2D and 3D electrohydrodynamic convection instability with crossflow1, YIFEI GUAN, JAMES RILEY, IGOR NOVOSSELOV, University of Washington — The study focuses on the electro-hydrodynamic (EHD) instability for flow between to parallel electrodes with unipolar charge injection with crossflow. Lattice Boltzmann Method (LBM) with two-relaxation time (TRT) model is used to study flow pattern [1]. Under strong charge injection and high electrical Rayleigh number, the system exhibits electroconvective vortices. Disturbed by different perturbation patterns, the flow patterns develop according to the most unstable modes. The unstable modes are obtained by dynamic mode decomposition (DMD) on the transient numerical solutions. Once the steady-state solution is obtained, Couette and Poiseuille cross-flow are applied. The flow patterns change according to the strength and direction of the cross-flow. When the cross-flow velocity is greater than a threshold value, in the 2D scenario, the vortices are suppressed [2], and in 3D the instability flow patterns would develop into streamwise rolling patterns. References: [1] Y. Guan and I. Novoselov, Two Relaxation Time Lattice Boltzmann Method Coupled to Fast Fourier Transform Poisson Solver: Application to Electroconvecive Flow, Journal of Computational Physics (accepted for publication) (2019). [2] Y. Guan and I. Novoselov, Numerical Analysis of Electroconvection Phenomena in Cross-flow, arXiv preprint arXiv:1812.10899 (2018).

1This research was supported by DHS Science and Technology Directorate, and UK Home Office and by the National Institutes of Health.
9.18AM C36.00007 A Front Tracking Method for the Investigation of the Electrohydrodynamic Instability Between Two Immiscible Fluids, ILKE KAYKANAT, Bogazici University, METIN MURADOGLU, Koc University, KEREM UGIZ, Bogazici University — When an electric field is applied to the flat interface between two immiscible liquids flowing in a microchannel, the interface can be deflected and microdroplets can be obtained. In this study, a front-tracking method is presented for direct numerical simulations of two-phase systems to study the effects of the electric field applied normal to the flat interface between two immiscible, Newtonian fluids. The method is developed using a one-field formulation of the governing equations. The interface is tracked explicitly using a Lagrangian grid and the flow equations are solved on a fixed Eulerian grid. The effects of the applied voltage, the viscosity, and the base-flow strength on the nonlinear evolution of the interface are studied. Furthermore, in this study, the results obtained by the interface tracking method will be compared with the results obtained by both long-wave analysis and the experimental results obtained in our group.

1BAP Project No: 15441

9.31AM C36.00008 Electroconvection near a metal electrode surface, GAJOIN LI, LYNDEN ARCHER, DONALD KOCH, Cornell University — Electroconvection in electrodeposition leads to fast dendrite growth undermining the efficacy of the process. Above a critical voltage, the electrohydrodynamic instability generates an electro-osmotic slip velocity at the edge of the space charge layer, creating convective flows in the electrolyte which cause an overlimiting current and a nonuniform ion flux. The nonuniform deposition of cations on a metal anode and its coupling with the electroconvection cause fast dendrite growth and lead to premature cell failure. The deposition rate of cation on the metal surface, which is described by the well-known Butler-Volmer equation, determines the ion transport below the limiting current. However, how does it affect the electroconvection in the overlimiting regime is not clear yet. In this work, we use the stability analysis and direct numerical simulation to investigate the linear instability and the dynamics of the electroconvection.

9.44AM C36.00009 Surface Reaction Driven Flow, ABIMBOLA ASHAJU, JEFFERY WOOD, ROB LAMMERTINK, University of Twente — Bimetallic nanorods in form of microswimmers within an aqueous solution exhibits self-propulsion that is powered by self-electrophoresis. This bimetallic catalytic system can be immobilized to generate convective flow thereby acting as a micropump. In this work, we focus on experimental and numerical analysis that provides fundamental insight on the key elements including the generated electric field, reaction kinetics and diffusion-electroosmotic phenomena that control the resulting mass transport characteristics in these systems. The catalytic current between the electrodes and the induced potential that governs the reactive fluxes are measured electrochemically, proton concentration gradients originating from the catalytic reaction are imaged and quantified using fluorescence lifetime imaging, while the fluid flow is visualized with 3D particle tracking. Numerical simulations reveals the interplay of electrodes surface reactivity pattern represented by the dimensionless Damkohler number, with the electrokinetic phenomena that controls the release and depletion of protons and consequently the resulting induced fluid flow. This work highlights the ability of surface induced convective fluid flow in electrochemical systems to reduce mass transport limitations.

9.57AM C36.00010 Experimental and numerical determination of flow mixing enhancement in electrokinetics cross shaped microchannel flows due to spatially periodic wall perturbations, AMADOR GUZMAN, MARIO DICAPUA, DAMING CHEN, FRANCISCO MONTERO, MARIA CANALES, Pontificia Universidad Catolica de Chile, LABONACHIP TEAM — Liquid flows in microchannels usually occur at very low Reynolds numbers (<1) because inertial forces are strongly dumped by the very small microchannel characteristic length and the dominant viscous forces. When an electrical field is applied to the microchannel ends, a time-dependent stable flow arises at a very low Reynolds number because another non dimensional parameter, the Rayleigh number, takes a dominant role. When the Rayleigh numbers surpass a threshold value, a unstable behavior is obtained, the flow becomes chaotic and consequently leading to an increase of the fluid mixing that is particularly important when liquids with different electrical conductivities need to be mixed, particularly in physiological flows. One way of achieving this type of behavior but to a lower critical Rayleigh number is by adding spatially periodic inhomogeneities to the channel walls. In this research, we investigate the flow mixing enhancement occurring in a cross-shaped microchannels when liquids with different electrical conductivities flow through the microchannel at different inlets and flows downstream due to the application of an electrical field and get together in the outlet branch of the microchannel, which contain spatial periodic wall inhomogeneities. Experiments and numerical simulations are carried out to determine and evaluate stable and unstable behaviors and evaluate the flow mixing enhancement.

Sunday, November 24, 2019 8:00AM - 10:10AM
Session C37 Multiphase Flows: Turbulence 619 - Antonino Ferrante, University of Washington

8:00AM C37.00001 Wavelet-spectral analysis of droplet-laden isotropic turbulence, ANDREAS FREUND, ANTONINO FERRANTE, University of Washington — The energy spectrum for homogeneous isotropic turbulence is computed using the Fourier transform of the velocity field. In the case of multiphase turbulent flows, the velocity field is non-smooth at the interface between the carrier fluid and the dispersed phase, so the Fourier energy spectra exhibit spurious oscillations at high wavenumbers. An alternative definition of the spectrum uses the wavelet transform, which can handle discontinuities locally while additionally preserving spatial information about the field. We propose using the wavelet energy spectrum to study multiphase turbulent flows and a new decomposition of the wavelet energy spectrum into three contributions corresponding to the carrier phase, droplets, and interaction between the two. Lastly, we apply the new wavelet-decomposition tools in analyzing the DNS data of droplet-laden decaying isotropic turbulence. Our results show that, in comparison to the spectrum of the single-phase case, the droplets (i) do not affect the carrier-phase energy spectrum at high wavenumbers, (ii) increase the total energy spectrum at high wavenumbers by increasing the interaction energy spectrum at these wavenumbers, and (iii) decrease the total energy at low wavenumbers by increasing the dissipation rate at these wavenumbers.

8:13AM C37.00002 Direct numerical simulation of droplet-laden homogeneous shear turbulence, PABLO TREFZTZ-POSADA, ANTONINO FERRANTE, University of Washington — We have performed direct numerical simulations (DNS) of droplet-laden homogeneous shear turbulence (DLHST) at initial \( Re_\text{D} = \frac{1}{50} \) with 6260 droplets of diameter approximately equal to the Taylor lengthscale (i.e., 5% droplet volume fraction). The droplet to carrier-fluid density and viscosity ratios have been set to 10. The droplet Weber number based on the r.m.s. velocity has been varied between \( 0.1 \leq We_{\text{rms}} \leq 5 \). First, we present our numerical methods for overcoming the challenges of simulating DLHST. Then, we present the effects of varying the shear number (\( Sh \)) on the budget of turbulence kinetic energy (TKE). For example, in the two-fluid TKE equation the power of the surface tension is directly proportional to the rate of change of droplet surface area (with opposite sign). DNS results show the effects of shear on droplet deformation/breakup/coalescence and how this affects the power of the surface tension, and, thus, the evolution of TKE.
8:26AM C37.00003 Wavelet analysis of spectral energy transfer in two-way coupled particle-laden turbulence. MIRALIREZA NABAVI BAVIL, Arizona State University, MARIO DI RENZO, Center for Turbulence Research, Stanford University, Stanford, CA 94305, USA, JOEL INGLAE KIM, Arizona State University. This study investigates the effects of two-way coupling between the carrier and dispersed phases on the spectral transfer of turbulent kinetic energy (TKE) using the wavelet multiresolution analysis (WMRA). Direct numerical simulations of decaying homogeneous isotropic turbulence laden with inertial particles are performed at Stokes numbers $St = 0, 1$ and $10$ with the point-particle assumption. TKE decreases monotonically as $St$ increases, while this is not the case for the dissipation rate. A multiresolution analysis based on orthonormal wavelet transform is developed to evaluate spectral energy transfer due to different physical mechanisms near particle clouds interacting in two ways with the carrier-phase turbulence. Spectral statistics conditioned on the Eulerian particle number density are examined to understand the physical trends of spectral energy transfer as a function of Stokes number and discuss their significance in modeling.

8:39AM C37.00004 Cascades of bubbles in turbulent breaking waves. WAI HONG RONALD CHAN, PERRY JOHNSON, JAVIER URZAY, PARVIZ MOIN, Center for Turbulence Research, Stanford University. Turbulent breaking waves entrain air cavities that break up and coalesce to form polydisperse clouds of bubbles. A bubble-tracking algorithm is developed to identify bubble breakup and coalescence events in interface-capturing two-phase numerical simulations, and to quantify the resulting transfers of air between bubbles of different sizes. The evolution of the volume- and ensemble-averaged bubble size distribution resulting from imbalances of the averaged transfer fluxes is described by a population balance equation with models for breakup and coalescence kernels. The formalism resembles the phenomenology of the Richardson-Kolmogorov energy cascade in single-phase turbulence. In order to demonstrate the presence of a bubble cascade, the transfer of air mass in bubble-size space by breakup and coalescence is examined for an ensemble of simulations of turbulent breaking waves. For breakup, a quasi-local transfer is observed in which the net transfer of air across a certain bubble size primarily depends on the number and breakup frequency of bubbles of similar sizes. This quasi-locality suggests that the statistics of bubble breakup at intermediate sizes are largely independent of the smallest and largest bubbles, in support of the idea of a bubble breakup cascade. $1$

1Agency for Science, Technology and Research, Singapore

8:52AM C37.00005 Low pressure events of finite size bubbles in homogeneous isotropic turbulence. MEHEDI HASEN BAPPY, ALBERTO VELA-MARTIN, PABLO CARRICA, GUSTAVO BUSCAGLIA. None — The study of the behavior of bubbles in turbulent flow is fundamental to the understanding of many two-phase flow applications such as cavitation inception. As gas nuclei evolve in a turbulent flow, the pressure fluctuations can dip below the vapor pressure and trigger cavitation events. Pressure statistics along trajectories of finite bubbles in isotropic homogeneous turbulence is investigated using direct numerical simulations at two $Re = 150, 240$. A modified Maxey-Riley equation is solved to track bubbles of different sizes in the turbulence field. The results show that larger bubbles are more attracted by the vortex cores and spend longer times at low-pressure regions. This has significant impact on the PDF of the pressure experienced by the bubbles, as well as on the statistics of low-pressure events (i.e., average frequency, distribution of duration and inter-event delays). The effect of gravity on these statistics is also addressed. It is shown that gravity is a bubble transport mechanism that competes with flow induced pressure gradients, making the bubbles less sensitive to low-pressure vortex cores.

9:05AM C37.00006 Liquid film breakup induced by turbulent shear flow. MELISSA KOZUL, NTNU, PEDRO COSTA, KTH, JAMES R. DAWSON, NTNU, LUCIA BRANDT, KTH. Gas turbine engines commonly employ prefilming airlift atomizers for liquid fuel injection. Supplied from holes or slits upstream, the liquid fuel forms a thin film over the prefilming surface before being driven to the atomizing edge by a turbulent shear flow. The use of a second air stream on the other side of the prefilmer to prevent fuel accumulation means the breakup and eventual atomization of the liquid film occurs in the shear zone formed by the co-flowing air streams (Aigner & Wittig, J. Eng. Gas Turbines Power (1988) vol. 110, pp. 105 - 110). We consider a simplified numerical setup using a recently developed fluid model (Li et al., J. Comput. Phys. (2012) vol. 231, pp. 2328 - 2358) to simulate this multiphase problem. A liquid film is sandwiched between sheared turbulent gas flows from a precursor simulation, which serve to deform and then rupture the liquid film. The simplified setup allows us to systematically vary parameters such as film thickness and turbulent gas flow Reynolds number to gauge the effect upon momentum transfer into the initially stationary liquid film. Understanding and controlling the route to breakup and atomization of liquid fuels in such systems is of primary practical concern in modern gas turbine engine design.

9:18AM C37.00007 Turbulence modulations induced by a swarm of surfactant-laden droplets. GIOVANNI SOLIGO, ALESSIO ROCCON, ALFREDO SOLDATI, TU Wien; University of Udine. We use direct numerical simulations of turbulence coupled with a two-order-parameter phase-field method to describe the complex dynamics of a swarm of surfactant-laden droplets in turbulence. Two separate Cahn-Hilliard (CH) equations define the dynamics of the interface and of the local surfactant concentration. An interfacial term, based on the Korteweg tensor, accounts for the effect introduced by a surfactant-laden, deformable interface on the flow field. In particular, the interface term accounts for both normal (capillary) and tangential (Marangoni) stresses at the interface. The former originate from the presence of the interface, while the latter arise whenever surface tension gradients along the interface are present. These surface tension gradients originate from a non-uniform surfactant distribution. The presence of a surfactant-laden interface thus affect the local flow field and turbulence and modulates exchanges of momentum between the droplet fluid and the carrier fluid. Here we will present the effects of different reference (clean interface) surface tension values and different types of surfactant on these exchanges.

9:31AM C37.00008 Turbulence statistics in a negatively buoyant particle plume. EVAN VARJANO, ANKUR BORDOLOI, University of California Berkeley, CHRIS LAI, Los Alamos National Laboratory, LAURA CLARK, Stanford University, GERARDO VELIZ, Cornell University. Plumes containing bubbles, particles and droplets are found in many natural phenomena as well as industrial applications. We report herein the turbulence statistics in a negatively buoyant multiphase plume containing heavy particles. We generate the plume by continuously releasing nylon particles of size 2 mm inside a salt-water tank via screw-conveyor based release mechanism. The two phases are refractive-index matched, that enables us to measure the local velocity in the salt-water via stereoscopic particle image velocimetry. Besides some structural differences, the turbulence statistics in a particle plume resemble that measured in a bubble plume. The turbulent kinetic energy (TKE) production by particles (or bubbles) roughly balances the viscous dissipation, except near the plume centerline. We observe a ~3 power-law in the one-dimensional power-spectra of the velocity fluctuations that puts both the particle and bubble plume in a category different from single-phase shear-flow turbulence.
9:44AM C37.00009 Jammed emulsions via turbulent stirring. FEDERICO TOSCHI, IVAN GIROTTO, GIANLUCA DI STASO, KARUN DATADJEN, Eindhoven University of Technology, The Netherlands; ROBERTO BENZI, University of Rome Tor Vergata, Italy; PRASAD PERLEKAR, Tata institute of fundamental research, Hyderabad, India; ANDREA SCAGLIARINI, IAC-CNR, Rome, Italy — Stabilised dense emulsions are common in many foods and cosmetics products (e.g. mayonnaise). Such complex fluids, made of two immiscible fluid components and a stabilizing agent (e.g. surfactant), behave like an elastic solid below a critical yield stress and flow like a viscous fluid above it. More generally, stabilized emulsions display all the rich phenomenology and rheology typical of soft-glassy materials. Stabilized emulsions are often produced via large-scale turbulent stirring. This raises the questions of e.g. how the emulsion structure depends on the turbulent stirring protocol and what are the rheological properties of the obtained emulsion. We employ large-scale 3d direct numerical simulations based on the multi-component Lattice Boltzmann method and second-bond coupling to numerically investigate turbulent emulsification. We show that turbulence is effective in producing a jammed state. We report the protocol followed in order to achieve packing fractions above 70% of the dispersed fluid phase and we characterize the yield stress of the obtained emulsions. In general, our model can be used to investigate catastrophic phase inversion, an event occurring either when the forcing intensity exceeds a threshold value or for excessive depletion of the matrix phase.

9:57AM C37.00010 Impact of turbulence on cloud microphysics of water droplets population. MINA GOLSHAN, FEDERICO FRATERNALE, MARCO VANNI, DANIELA TORDELLA, Politecnico di Torino — This work focuses on the turbulent shear-less mixing structure of a cloud/clear-air interface with physical parameters typical of cumulus warm clouds. We investigate the effect of turbulence on the droplet size distribution, in particular we focus on the distributions broadening and on the collision kernel. We performed numerical experiments via Direct Numerical Simulations (DNS) of turbulent interfaces subject to density stratification and vapor density fluctuation. Specifically, an initial super-saturation around 2% kinetic energy of 100 cm²/s² are set in the DNSs. The Taylors Reynolds number is between 150 and 300. The total number of particles is around 5-10 millions, matching an initial liquid water content of 0.8 g/m³. Through these experiments, we provide a measure of the kernel of collisional integral operators to be compared with literature models [Saffman Turner, 1955] and possibly used inside drops Population Balance Equations (PBE) that include both processes of drops growth by condensation/evaporation and aggregation.

Sunday, November 24, 2019 8:00AM - 10:10AM — Session C38 Geophysical Fluid Dynamics Sediment Transport 620 - Bruce Sutherland, University of Alberta

8:00AM C38.00001 Dispersion and transport of tracer particles at an upper ocean front. VICKY VERMA, SUTANU SARKAR, University of California San Diego — Turbulence resolving simulations of upper-ocean fronts show the formation of thin filament structures during the evolution of baroclinic instability. The filaments develop large vertical vorticity and vertical velocity; furthermore, long filaments roll up to form coherent submesoscale eddies. The role of coherent structures, i.e. the vortex filaments and the eddies, in vertical and lateral transport and dispersion of particles has been studied for a submesoscale mixed layer front using tracer particles which advect with the local fluid velocity. The time evolution of tracer particles reveals stirring across the front, as well as vertical transport directed by the coherent structures. There are vortex filaments on both the heavier and lighter sides of the front that provide direct pathways for vertical transport. The filament on the heavier side downwells the local (heavier) fluid particles to the bottom, spiraling around the eddy, while the filament at the lighter side upwells the local (lighter) fluid particles to the surface. Subsequently, the downwelled and upwelled particles slowly adjust to a smooth stable profile. The effect of coherent structures on the dispersion of particles is studied using single-particle, two-particle and multi-particle dispersion statistics.

8:13AM C38.00002 Sediment transport and morphodynamics induced by a translating vortex. MATIAS DURAN-MATUTE, SAMUEL GONZALEZ-VERA, GERTJAN VAN HEIJST, Eindhoven University of Technology — We present experimental results on the effect of a translating monopolar vortex on a sediment bed. Experiments took place inside a water-filled square tank with a flat layer of small spherical particles at the bottom and placed on top of a rotating table. A vortex is generated on the tip of a plate perpendicular to one of the sidewalls by drastically changing the speed of the rotating table from a state of solid body rotation. This change was designed such that a residual current remained, advecting the vortex away from the plate. Strong-enough vortices bring sediment into suspension and transport it along their path. As the vortices weaken, the sediment settles back into the bed. This mechanism produces changes in the bed. Measurements of the flow were obtained through particle image velocimetry (PIV). The region of suspended sediment was reconstructed using images of the particles illuminated by a vertically oscillating horizontal laser sheet, and the changes in the bed thickness were measured with a photogrammetric technique. The strength of a vortex is the main parameter governing the capture and suspension of particles with similar settling velocity. A power law was found between the vortex strength and the net displaced particle volume.

8:26AM C38.00003 Two-way coupled direct numerical simulations of sinuous-crested bedforms. NADIM ZGHEIB, School of Engineering - Lebanese American University, S BALACHANDAR, Dept. of Mechanical and Aerospace Eng. - University of Florida — We present results from direct numerical simulations on the transition from straight-crested to sinuous-crested bedforms. The numerical setup is representative of turbulent open channel flow over an erodible sediment bed at a shear Reynolds number of Re₉ = 180. The immersed boundary method accounts for the presence of the sediment bed. The simulations are two-way coupled in the sense that the turbulent flow can erode and modify the sediment bed, and in turn, the sediment bed modifies the overlying flow. The coupling from the flow to the sediment bed occurs through the Exner equation, while the back coupling from the bed to the flow is achieved by imposing the no-slip and no-penetration boundary condition at the immersed boundary. The simulation setup is similar to that by Zgheib et al. (https://doi.org/10.1002/2017JF004398) except for the presence of sidewalls to better mimic laboratory flume conditions. Sidewalls are observed to significantly increase bedform sinuosity. We also investigate the effect of domain size and a zero sediment flux boundary condition on crestline sinuosity.

1Support from ExxonMobil Upstream Research Company through grant no. EM09296
8:39AM C38.00004 Dune-dune repulsion

1. KAROL BACIK, University of Cambridge, SEAN LOVETT
2. Schlumberger Cambridge Research, COLM-CILLE CAULFIELD, NATHALIE VIPREND, University of Cambridge — Dunes are coherent sedimentary structures which arise spontaneously due to the dynamical interplay between granular matter and the flow of the overlying fluid. Natural dunes rarely occur in isolation. Aeolian dunes form vast dune fields and subaqueous bedforms occur in groups. As of now, the mechanisms which regulate the large scale organisation of dune fields are poorly understood. In particular it is unclear if the dune configurations we observe are stable or transient. Here we investigate the dynamics of a quasi-2D dune corridor using a subaqueous experiment in an annular geometry. We show that the corridor structure appears to be robust and that stabilisation is achieved by long range dune-dune interactions. Our experiments reveal that by altering the flow, dunes strongly affect the shape and the migration rate of their downstream neighbours which leads to an effective dune-dune repulsion. Here we discuss the physical origin of the repulsion mechanism and explore its consequences for the system-level dynamics of the dune corridor.

1. KB acknowledges a PhD studentship from Schlumberger Cambridge Research, NV was supported by a Royal Society Dorothy Hodgkin Fellowship (DHI12012)
2. Author no longer at the institution

8:52AM C38.00005 Aggregate particle settling from a laboratory river plume

DAVID DEEPWELL, BRIANNA MUELLER, BRUCE SUTHERLAND, University of Alberta — Motivated by understanding how particles in sediment bearing river plumes ultimately settle, we present laboratory experiments of a constant-flux fresh water surface gravity current containing a low (< 1%) concentration of dense particles which initially overrides a saline ambient. Two classes of experiments were performed: 1) with a uniform density saline ambient fluid and 2) a two-layer ambient fluid. In both classes of experiments, collective settling combined with ambient recirculating flows result in particles settling out predominantly as a large single vertical plume. As the plume falls, the centre of the plume travels faster than the fluid around it causing an entrainment of neighbouring particles leading to a narrowing of the plume with depth. In a two-layer stratification with micrometer-sized particles, the settling is delayed at the interface between the upper and lower layers as a result of lighter fluid being dragged down with each particle in its surrounding viscous boundary layer thereby decreasing its effective density. The width and location of the sedimentation plume as well as the enhanced settling velocity of particles in the plume is characterized with respect to particle density and concentration.

9:05AM C38.00006 Strain-hardening evolution on sediment transport under cyclic laminar shear

1. FERNANDO CEZ, ERICK M. FRANKLIN, University of Campinas (UNICAMP), MORGANE HOUSSAIS, Levich Institute, City College of CUNY, PAULO E. ARRATIA, DOUGLAS J. JEROLMACK, University of Pennsylvania — Fluid-driven granular flows are common in nature and industry, yet difficult to understand. We employ a large eddy simulation (LES) code with micrometer-sized particles sheared by a laminar flow; this allows visualization of particle transport throughout the bed. We introduce stepped cycles of fluid shear and examine the downward propagation of an “unjamming” front, which marks the boundary between a granular fluid and a creeping solid. Repeated shear cycles give rise to strain hardening, as the fluidized layer thins over progressively longer times. Such experimental results are compared to numerical simulations: isotropic grain fabric structures aligned with the applied shear. When shear direction is reversed, the bed recovers some (but not all) of its initial mobility. Strain hardening, associated compaction and grain-fabric formation, occur in the observed creep regime. Beyond a critical shear rate, grains are fluidized resulting in dilation and destruction of granular fabrics. Results explain puzzling observations of bed-load hysteresis in natural rivers and delineate the transport regimes under which memory is created and destroyed in sedimentary beds.

1. FAPESP (2016/18189-0, 2018/23838-3, 2018/14981-7), Army Research Office (W911-NF-16-1-0290)

9:18AM C38.00007 LES of a large-scale river with arbitrarily complex bathymetry and wall-mounted structures

KEVIN FLORA, ALI KHOSRONEJAD, Stony Brook University — High-fidelity numerical simulation of flow in natural riverine environments is essential for understanding the potential impacts of flood events on infrastructure and the potential morphological changes to the channel. However, due to the bathymetric complexity and unique shapes of man-made structures located in the flow, it is very challenging to create high-fidelity models which accurately depict the dominant three-dimensional flow structures. Such flow dynamics often lead to the erosion of the banks, scour of infrastructure foundations and overall transport of sediment in the flow. As a result, simplified models are often used in engineering practice, but these models fail to accurately simulate important features of flow. We employ a large eddy simulation (LES) code to model flow around four bridge piers in the American River in California to study the qualitative and quantitative differences in the resulting data. Specifically, this study compares the results of the LES code with more simplified hydrodynamic models to assess the difference in the estimated bed shear stress distribution and the overall impacts of the bridge structures on the turbulence.

9:31AM C38.00008 Fluid-driven transport of spherical sediment particles: from discrete simulations to continuum modeling

QIONG ZHANG, KEN KAMRIN, Mechanical Engineering, Massachusetts Institute of Technology, ERIC DEAL, TAYLOR PERRON, EAPS, Massachusetts Institute of Technology, JEREMY VENDITTI, Simon Fraser University, SANTIAGO BENAVIDES, MATTHEW RUSHLOW, EAPS, Massachusetts Institute of Technology — Empirical bedload transport expressions commonly over- or underpredict sediment flux by more than a factor of two, even under controlled laboratory conditions. In this work, a novel Element Method and Lattice Boltzmann Method are coupled together to simulate 3D fluid-driven transport problems, in which the spherical sediment particles are fully resolved. After comparisons with flume experiments are made to test the numerical simulations, the grain-scale physics is studied, such as the role of the gravitational forces and other order descriptions of the grain motion. A more robust continuum model, unifying empirical models under various conditions and in different regimes, is further proposed based on the new grain-scale understanding of the mechanisms.

9:44AM C38.00009 Three-phase flow LES of flash floods in a real-life desert stream

1. ALI KHOSRONEJAD, KEVIN FLORA, Stony Brook University, ALI KHOSRONEJAD TEAM — We present a fully coupled three-phase flow model of air-water-sediment to simulate numerically the propagation of flash floods in field-scale dry-bed desert streams. The turbulent flow and free surface of the flash flood are computed using large-eddy simulation (LES) level-set method, respectively. The evolution of stream morphology, due to the propagating flood on the mobile bed, is calculated using an Eulerian morphodynamics model based on the curvilinear immersed boundary method. We demonstrate the capabilities of our numerical framework by applying it to simulate a flash flood event in a 0.65 km-long reach of a desert stream in California. The simulation result of the stream includes a number of bridge foundations. Simulation results of the model for the flash flood event revealed the formation of highly complex flow field and scour patterns within the stream. Moreover, our simulation results show that most of the scour processes takes place during the steady phase of the flash flood. The transient phase of the flash flood is rather short and contributes to a very limited amount of erosion within the desert stream. Acknowledgment: support for this work is provided by NSF award EAR-1823121.
9:57AM C38.00010 Euler-Lagrange Simulations of Bedload Transport and Bedform Generation1, LIHENG GUAN, JORGE SEBASTIAN SALINAS, S. BALACHANDAR, University of Florida — Bedload transport is a subset of sediment transport in which the sediment particles rolls, slides and saltates under the influence of the overlying flow field. The particles in the bedload occasionally gets lifted and gets suspended at high enough flow rate. In this work, the bedload transport problem is investigated by performing numerical simulations with implementation of Euler-Lagrange point-particle approach. The fluid field is solved on the Eulerian reference frame while the motion of sediment particles are tracked on the Lagrangian reference frame. Moreover, the collisions between particles are handled using a soft-sphere collision model which includes both a normal collision force and a tangential collision force. Additionally, the rotational motion of particles is also considered in this work. In the bedload transport simulation, a fully developed turbulent channel flow is imposed over an initially flat bed. Various bedform structures, from longitudinal streaks to ripples, arise as the sediment bed evolves. The results are compared with those using Exner’s equation.

1We acknowledge ExxonMobil for their support of this work.

Sunday, November 24, 2019 10:38AM - 12:38PM – Session D01 Awards Session: Presentation of Awards and DFD Fellowships (Otto LaPorte Lecture, Stanley Corrsin Award) 6abc - Detlef Lohse, University of Twente: Karen Daniels, NC State University

10:38AM D01.00001 PRESENTATION OF AWARDS AND DFD FELLOWSHIPS

11:08AM D01.00002 Otto LaPorte Lecture: Experiments in High Reynolds Number Flows1, ALEXANDER SMITS, Princeton University — To attain a very large range of Reynolds numbers in the laboratory, it is convenient to use high-pressure air. We have made extensive use of this approach to study the behavior of full-developed pipe flow, turbulent boundary layers, and the wakes downstream of bodies-of-revolution. I will summarize some of the major results obtained for the pipe and boundary layer, including the scaling of the mean velocity profile, the streamwise turbulence intensity, and the spectra. I will also discuss some present and future directions of this research, which largely focus on high Reynolds number non-canonical flows.

1This work has been supported by ONR, most recently through Grant N00014-17-1-2309.

11:53AM D01.00003 Stanley Corrsin Award Talk: From Microstructure to Models – Fluid Mechanics of Suspensions1, JEFFREY F MORRIS2, Levich Institute, CUNY City College of New York — Suspensions of particles in Newtonian liquids are simple complex fluids. Describable by just a handful of macroscopic variables, suspensions are in one sense simple; but the microscopic state is responsive to variation of these variables, making the flow properties non-Newtonian, and hence these are complex fluids. Our work is motivated by the scientific goal of developing a fluid mechanics of the bulk suspension behavior, toward a framework allowing analysis and computation; the hope is that this will expand understanding of suspensions found in engineering and the environment. Progress toward this goal often benefits from examination of the microscopic mechanics. Particle migration and its basis in rheological properties will be discussed, along with the microstructure of particles leading to these properties. Phenomena in extremely dense suspensions – shear thickening and jamming – will be discussed with an emphasis on how these motivate a shift in focus, from the particles to the force networks developed under flow. The prior Stokes-flow examples will be complemented by presentation of inertial transitions observed in Taylor-Couette flow of suspensions, to illustrate challenges faced in developing a true fluid mechanics of these mixtures.

1NSF 1605283
2

Sunday, November 24, 2019 2:03PM - 2:38PM – Session E02 Invited Talk: Advancing understanding of turbulence through extreme-scale computation 6b - Katepalli Sreenivasan, NYU

2:03PM E01.00001 Understanding of turbulence through extreme-scale computation1, P.K YEUNG, Georgia Institute of Technology — Since its beginnings in the early 1970s, direct numerical simulation of turbulence in canonical geometries has always been the computational approach that is physically the most accurate, a massive source of data, and a grand challenge in high-performance computing, whose definition has evolved by many orders of magnitude since then. In particular, challenges driving the need for ever-larger simulations on a 3D periodic domain include, to name a few, fine-scale intermittency at high Reynolds number, with stringent resolution requirements, the mixing of a passive scalar of low molecular diffusivity, and the motion of fluid and inertial particles in a Lagrangian framework. Most known state-of-the-art simulations have employed massive CPU-based parallelism, which is ultimately limited by communication costs traceable to the multi-dimensional nature of the Navier-Stokes equations. However, current trends in pre-Exascale leadership computing are pointing to the growing importance of heterogeneous platforms, whose principal advantage is accelerated computation and whose full exploitation requires a new paradigm in code development. In this talk, we will discuss the major features of a new pseudo-spectral code which has been shown to scale satisfactorily up to a problem size of 184323 resolution on the currently world’s fastest GPU-based “Summit” supercomputer located at the Department of Energy Oak Ridge National Laboratory. We show that new simulations at this resolution are enabling significant advancements in studies of small-scale turbulence, with emphasis on extreme events where fluctuations of energy dissipation rate and enstrophy (vorticity squared) can reach O(103) times of the mean value or higher. We also discuss briefly how recent simulations at resolution 81923 or higher are contributing to progress in the study of turbulent mixing, for both passive and active scalars in non-unity Schmidt number regimes.

1Supported by DOE (INCITE and Summit Early Science Programs at OLCF), NSF (PRAC at NCSA and TACC)

Sunday, November 24, 2019 2:03PM - 2:38PM – Session E02 Invited Talk: Living flow networks 6c - Peko Hosoi, MIT

2:03PM E01.00001 Understanding of turbulence through extreme-scale computation1, P.K YEUNG, Georgia Institute of Technology — Since its beginnings in the early 1970s, direct numerical simulation of turbulence in canonical geometries has always been the computational approach that is physically the most accurate, a massive source of data, and a grand challenge in high-performance computing, whose definition has evolved by many orders of magnitude since then. In particular, challenges driving the need for ever-larger simulations on a 3D periodic domain include, to name a few, fine-scale intermittency at high Reynolds number, with stringent resolution requirements, the mixing of a passive scalar of low molecular diffusivity, and the motion of fluid and inertial particles in a Lagrangian framework. Most known state-of-the-art simulations have employed massive CPU-based parallelism, which is ultimately limited by communication costs traceable to the multi-dimensional nature of the Navier-Stokes equations. However, current trends in pre-Exascale leadership computing are pointing to the growing importance of heterogeneous platforms, whose principal advantage is accelerated computation and whose full exploitation requires a new paradigm in code development. In this talk, we will discuss the major features of a new pseudo-spectral code which has been shown to scale satisfactorily up to a problem size of 184323 resolution on the currently world’s fastest GPU-based “Summit” supercomputer located at the Department of Energy Oak Ridge National Laboratory. We show that new simulations at this resolution are enabling significant advancements in studies of small-scale turbulence, with emphasis on extreme events where fluctuations of energy dissipation rate and enstrophy (vorticity squared) can reach O(103) times of the mean value or higher. We also discuss briefly how recent simulations at resolution 81923 or higher are contributing to progress in the study of turbulent mixing, for both passive and active scalars in non-unity Schmidt number regimes.

1Supported by DOE (INCITE and Summit Early Science Programs at OLCF), NSF (PRAC at NCSA and TACC)
There is a wide range of highly important applications, not least the leads to the enhanced irreversible transport of heat and other scalars (such as salt and pollutants) in density-stratified fluids is a fundamental problem. What we do not know?

2:03PM E02.00001 Living flow networks, ELENI KATIFORI, University of Pennsylvania — Complex life larger than a humble nematode would not be possible without a circulatory system. Plants, fungi, and animals have developed vascular systems of striking complexity to solve the problem of long distance nutrient delivery, waste removal, and information exchange. These biological flow systems are frequently not static, but dynamically alter the diameters of their vessels in order to optimally achieve their intended function. The conductance of these vessels does not have to be linear, although it is frequently treated as such. Inspired by haemodynamic oscillations in the cortex, in this talk we explore how a superficially simple system of laminar flow vessels can sustain dynamic local oscillations, even in the absence of time varying boundary conditions. We will show how these non-linear conducting vessels redirect high volumes of flow to different locations of the network. We will present how the dynamics of this system is akin to an excitable system without any explicit refractory period in the vessel response. Last, we will discuss how we can harness the interplay of network topology and fluid dynamics to control flow in unexpected ways, and what this dynamic flow network behaviour can tell us about living systems.

Sunday, November 24, 2019 2:03PM - 2:38PM –
Session E03 Invited Talk: Lagrangian turbulent thermal convection 6e - Nicolas Mordant, LEGI, University Grenoble-Alps

2:03PM E03.00001 Lagrangian turbulent thermal convection1. MICKAL BOURGOIN, Physics Laboratory, CNRS / ENS de Lyon — Thermally driven turbulence plays an important role in numerous areas of science and technology. Many natural (atmospheric and oceanic dynamics, processes in planetary cores, etc.) and industrial (cooling of buildings, heat exchangers, etc.) flows are indeed controlled by thermal convection, and often exhibit peculiar inhomogeneity and non-stationnary behavior. While a prolific literature exists regarding transport phenomena for homogeneous, isotropic and stationary turbulence, understanding the transport properties of thermal turbulence remains a real challenge, especially from the Lagrangian prospect. In this presentation, recent experimental results on the Lagrangian description of thermal turbulence in Rayleigh-Bnard convection will be presented. The specific roles of small-scale turbulence and large scale inhomogeneities and unsteadiness will be discussed, revisiting both the Lagrangian dynamics of single particles and the celebrated pair dispersion problem.

Sunday, November 24, 2019 2:43PM - 3:18PM –
Session F02 Invited Talk: Reduced Order Modeling of Blood Flow 6c - Rajat Mittal, Johns Hopkins University

2:43PM F02.00001 Reduced Order Modeling of Blood Flow, SHAWN SHADDEN, UC Berkeley — Image-based blood flow modeling has become a cornerstone in cardiovascular research and has recently gained regulatory approval and clinical adoption for diagnosis of cardiovascular disease. This technology combines medical imaging with advanced computational fluid dynamics modeling to predict flow and pressure in regions of the cardiovascular system. Toward the same goal, I will discuss combining medical imaging with simplified, reduced-order modeling (ROM) of blood flow. These ROM developments enable new possibilities to use modeling for timely decision support in clinical settings, and improve the abilities to perform data assimilation, optimization, parameter tuning, uncertainty quantification and sensitivity analysis in cardiovascular modeling applications. I will discuss the underlying methodologies, and comparison of our ROM approaches against current states-of-the-art.

Sunday, November 24, 2019 2:43PM - 3:18PM –
Session F03 Invited Talk: Open questions in turbulent stratified mixing: Do we even know what we do not know? 6e - Peter Diamessis, Cornell University

2:43PM F03.00001 Open questions in turbulent stratified mixing: Do we even know what we do not know? C. P. CAULFIELD, BP Institute & DAMTP, University of Cambridge — Understanding how turbulence leads to the enhanced irreversible transport of heat and other scalars (such as salt and pollutants) in density-stratified fluids is a fundamental and central problem in geophysical and environmental fluid dynamics. There is a wide range of highly important applications, not least the description and parameterization of diapycnal transport in the world’s oceans, a key area of uncertainty in climate modelling. Recently, due not least to the proliferation of data obtained through direct observation, numerical simulation and laboratory experimentation, there has been an explosion in research activity directed at improving community understanding, modelling and parameterization of the subtle interplay between energy conversion pathways, turbulence, and irreversible mixing in density-stratified fluids. However, as I will discuss in this talk, there are still leading order open questions and areas of profound uncertainty concerning turbulent stratified mixing. Therefore, I will present a personal perspective on some priorities for further research into this hugely complex, important and fascinating fluid dynamical challenge.
3:48PM G01.00001 Collapse of multiple holes in an unbounded liquid film, MICHAEL EIGENBROD, STEFFEN HARDT, Institute for Nano- and Microfluidics, TU Darmstadt — We study experimentally and theoretically the collapse of multiple holes in a liquid film. The time evolution of an ensemble of holes is examined using high-speed videomicroscopy and characterized by the time-dependence of the hole shapes. An analytical formula for the potential energy difference between an unbounded collapse of multiple holes in a liquid film. The time evolution of a multi-hole arrangement in a highly viscous film can be predicted through steepest descent of the potential energy in a confinement space representing the shapes of the holes. The theoretical model for the time evolution of the system if confirmed by experimental results for the collapse of multiple nearly circular holes.

4:01PM G01.00002 Thermal rupture of a free liquid sheet, JENS EGGERS, University of Bristol, GEORGY KITAVTSEV, University of Oxford, MARCO FONTELOS, Instituto de Ciencias Matematicas — We consider a free liquid sheet, taking into account the dependence of surface tension on the temperature or concentration of some pollutant. The sheet dynamics are described within a long-wavelength description. In the presence of viscosity, local thinning of the sheet is driven by a strong temperature gradient across the pinch region, resembling a shock. As a result, for long times the sheet thins exponentially, leading to breakup. We describe the quasi-one-dimensional thickness, velocity and temperature profiles in the pinch region in terms of similarity solutions, which possess a universal structure. Our analytical description agrees quantitatively with numerical simulations.

4:14PM G01.00003 Optimal Capillary Breakup Rheometer Procedures for Newtonian Filaments, SUBRAMANIAM BALAKRISHNA, WILLIAM SCHULTZ, University of Michigan (Ann Arbor, MI, US) — The differential analysis of McCarron et al (2016) determines the surface tension to viscosity ratio from the symmetry point of an unsteady Newtonian film profile and its derivatives. The experimental challenges are twofold: (a) the fourth derivative of the free surface radius is required and difficult to extract from low-resolution, pixelated and possibly noisy images and (b) the sensitivity of the surface tension to viscosity ratio to the stretch history. In particular, for filament evolution dominated by viscosity and surface tension, stretching too quickly makes the free surface profile nearly cylindrical, while stretching too slowly yields a quasi-static profile with no viscous information. We give strategies that optimize the ratio measurement including use of higher-order finite difference stencils and measurements made during stretch.

4:27PM G01.00004 ABSTRACT WITHDRAWN

4:40PM G01.00005 Interfacial perturbation classification of a fluid impulsively driven through a honeycomb mesh into a gas-filled cavity, TAKIAH EBBS-PICKEN, RUBERT MARTIN PARDO, McGill University, DAVID PLANT, General Fusion, ANDREW HIGGINS, JOVAN NEDIC, McGill University, MCGILL UNIVERSITY COLLABORATION, GENERAL FUSION COLLABORATION — We investigate the development of perturbations on a liquid-gas interface created by the impulsive motion of fluid through a honeycomb mesh. The effects of implosion speed, mesh geometry, angular velocity of the interface, and initial liquid depth relative to the mesh were experimentally investigated to determine their effects on the geometry of the perturbations formed. An experimental arrangement was created that allowed for the visualization of the cavity surface, and a classification based on the perturbation geometry was developed. Five different perturbations were observed, which were classified as follows: none, wavy, sharp, jetting and complex. The results showed that the parameter which had the largest impact on the initial perturbation growth was the fill depth of the liquid relative to the mesh, while the angular velocity and implosion speed had limited effects.

4:53PM G01.00006 The natural breakup length of a steady capillary jet, SASA BAJT, HENRY CHAPMAN, MAX WIEDORN, JURAJ KNOSKA, YANG DU, Center for Free-Electron Laser Science, Deutsches Elektronen Synchrotron (DESY), Notkestrasse 85, 22607 Hamburg, Germany, MICHAEL HEYMAN, Dept. Physics, University of Hamburg, Luruper Chaussee 149, 22761 Hamburg, Germany, BRAULIO GANAN-RIESCO, Ingenieria Tecnologias S.L., 41000 Carac, Sevilla, Spain, JOSE M. LOPEZ-HERRERA, MIGUEL A. HERRADA, FRANCISCO CRUZ-MAZO, ALFONSO M. GANAN-CALVO, Universidad de Sevilla, ETSI, 41092 Sevilla, Spain — The averaged natural breakup length of capillary jets ejected in inactive environments are determined by the liquid properties, its velocity and its diameter. Despite its theoretical and applied interest, a general procedure to predict that length has not been proposed yet. Here we describe and quantify the energy route that sets it. We find that the underlying mechanism that determines that length is the short-term transient growth rate of perturbations excited by the jet breakup itself. We propose a perturbation analysis of the time averaged energy conservation equation in the absence of body forces. The balance of total energy rates due to the perturbations is reduced, by dimensional analysis, to a closed algebraic expression with two universal constants. These constants are calculated by optimal fitting of a large set of experiments from diverse sources, experimental and numerical, which confirm the universal scaling law found.

5:06PM G01.00007 Modelling of perturbations in the surface of a cylindrical imploding rotating cavity, RUBERT MARTIN PARDO, McGill University, DAVID PLANT, General Fusion, JOVAN NEDIC, McGill University, GENERAL FUSION COLLABORATION — A cylindrical cavity is formed by rotating a liquid to solid body rotation resulting in a cylindrical liquid shell surrounding a gas-filled cavity. As this cavity is radially collapsed by pushing the fluid through a honeycomb mesh, perturbations begin to form on the interface. The dynamics of this perturbation can be well modelled by a second-order ODE and, as such, the initial velocity and amplitude of the perturbation have a decisive role on the exterior behavior of the roughness of the surface during the implosion. It will be shown how this initial perturbation, in turn, depends on different control parameters, namely the driving pressure profile, the rotation rate of the system, the initial liquid depth and the mesh geometry. A model is developed to account for this relation. The cavity surface was measured using high-speed videography and surface tracking digital techniques for a range of values of the control parameters sufficient to capture the change from a rough cavity surface to a smooth interface.

1NSERC Canada

1Supported by National Science Foundation Fluid Dynamics Program

1Funding from NSERC Canada

1Supported by the Ministerio de Economia y Competitividad (Spain), Plan Estatal Retos, project DPI2016-78887-C3-1-R
Grants W911NF1410301 and W911NF16C0117.

Expressions for temperature jumps and phase change rates are obtained from theoretical considerations and augmented by MD simulations.

Accounting for jump-conditions across phase interfaces using explicit interface tracking and discontinuous interpolation only at the interface.

Continuity/discontinuity. In this study, we explore the effect of temperature discontinuities at liquid-vapor interfaces before the system reaches a quasi-steady state condition; however, a dissimilar temperature profile was observed with the assumption of temperature entrainment minimization as well as fill ratio and optimized geometric dimensions for maximizing axial heat flow.

We exploit ideas otherwise applied to plastron respiration by aquatic insects. We thereby reject the popular simplification that the three-phase functioning of the heat pipe, the interface shape dictating the degree of capillary pumping that occurs. In formulating our mathematical model, this shape changes owing to a change of operating conditions. Far from being of academic interest, these details are essential to the effective shape change in total flow Q along the wedge is used to determine the mass flux from the liquid meniscus into the vapor due to phase change.

An imposed temperature gradient gives rise to Marangoni flow in the direction of decreasing temperature. The fluxes significantly in a quasi-steady state condition; however, a dissimilar temperature profile was observed with the assumption of temperature continuity/discontinuity. In the constrained vapor bubble system, designed by the group of J. Plawsky, liquid-vapor mixture inside an elongated cuvette of rectangular cross-section is subject to axial temperature gradients resulting in phase change and complex flow patterns. Experimental studies of this system have been conducted on the ground and onboard the International Space Station. A number of unexpected phenomena, most notably flooding of the hot end of the cuvette, were discovered in the microgravity experiment. In the present study, we focus on mathematical models designed to explain some of these phenomena via a study of corner flows driven by both Marangoni and an opposing capillary flow due to changes in temperature and curvature of the menisci respectively along the wedge. Also, phase change takes place at the liquid vapor interface and, depending on the location, can be either evaporation or condensation. A model is presented for a steady state corner wedge flow and compared with experimental observations. An imposed temperature gradient gives rise to Marangoni flow in the direction of decreasing temperature. The change in total flow Q along the wedge is used to determine the mass flux from the liquid meniscus into the vapor due to phase change.

This work was supported by NASA grant 80NSSC18K0332.

This work was supported by the Office of Naval Research Thermal Science Program, Award No. N00014-17-1-2767, and U.S. Army grants W911NF14140301 and W911NF16C0117.

4:27PM G02.00004 Liquid-Vapor Interface Configuration for Capillary Flow in a Heat Pipe. MUHAMMAD RIZWANUR RAHMAN, PRASHANT WAGHMARE, MORRIS R. FLYNN, University of Alberta — The shape of the liquid-vapor interface within a heat pipe wick is responsive to the wetting characteristics of the working fluid, wick geometry and other operational variables. This study critically investigates the underlying physics that dictate the interface shape and the manner in which this shape changes owing to a change of operating conditions. Far from being of academic interest, these details are essential to the effective functioning of the heat pipe, the interface shape dictating the degree of capillary pumping that occurs. In formulating our mathematical model, we exploit ideas otherwise applied to plastron respiration by aquatic insects. We thereby reject the popular simplification that the three-phase contact point is somehow anchored at a fixed elevation within the wick. Although multiple solutions are predicted, corresponding to different interface shapes, only one is mechanically stable and therefore physically-acceptable. When coupled with a complementary thermodynamic model, the unique interface shape can be predicted for prescribed operating conditions. Such information offers insights on the recess depth for entrainment minimization as well as fill ratio and optimized geometric dimensions for maximizing axial heat flow.
4:40PM G02.00005 Core Annular Flow Theory as Applied to the Adiabatic Section of Heat Pipes1, AISHWARYA RATH, MORRIS R. FLYNN, University of Alberta — Core annular flow theory is used to model the parallel flow of fluids of different phases and has been applied to industrial applications from bitumen hydrotransport to sub-aqueous drag reduction. Here we consider the extension of core annular flow theory to the study of the adiabatic section of heat pipes. Our aim is to develop a first-principles estimate of the conditions necessary to maximize the (counter) flow of liquid and vapor and, by extension, the axial flow of heat. Both planar and axisymmetric geometries are examined. Moreover, we consider heat pipes either containing or devoid of a wick. In these two respective cases, the peripheral return flow of liquid is driven by capillarity and by gravity. Our model is used to predict velocity profiles and the appropriate pressure gradient ratio (vapor-to-liquid). We further obtain estimates for the optimum thickness of the liquid layer. Note finally that when the liquid flow occurs via capillary pumping, there is a minimum surface tension below which the wick cannot supply a sufficient flow of liquid. We characterize this critical point in terms of e.g. the viscosity ratio, the density ratio and the wick depth, porosity and permeability.

1NSERC

4:53PM G02.00006 A non-Reynolds lubrication model and application to droplet levitation1, SHINTARO TAKEUCHI, JINGCHEN GU, Osaka University, AMARYU BARRAL, Tokyo University of Agriculture and Technology, Ecole polytechnique, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology, NRL COLLABORATION, LEVITATION COLLABORATION — An attempt for extending the applicability of the Reynolds lubrication theory is presented. By considering a larger surface-to-surface distance than that for the Reynolds lubrication theory, the effect of the non-negligible pressure gradient in the surface-normal direction is incorporated into the lubrication model. The analysis shows that the local pressure is separated into (i) a base component satisfying the Reynolds lubrication theory and (ii) an adjusting component varying in the surface-normal direction, and the second component is found to be related with the velocity of the local Couette-Poiseuille flow. The lubrication model is verified in a non-Reynolds regime of a flow between a moving curved object and stationary object, and good agreement in the pressure distributions by analytical and numerical methods is observed in both wall-normal and longitudinal directions. The lubrication model is also applied to a droplet levitation problem over a moving wall (Sawaguchi 2019) involving a µm-thin air film, and the levitation lift reproduced from the full drop profile is found to provide a more accurate and complete view of the levitation problem.

1This work is partly supported by Grant-in-Aid (B) Nos.16H04271 and 17H03174 and Grant-in-Aid (A) No.17H01246 of the Japan Society for the Promotion of Science (JSPS). One of the authors, A.B., acknowledges the Innovative Process & Materials led by iX - Ecole polytechnique and the Fondation de l'Ecole polytechnique and sponsored by SAINTGOBAIN.

5:06PM G02.00007 Population Balance Modelling and CFD analysis in a Multi-scale Approach for Flash Nano-precipitation1, ALESSIO LAVINO, Politecnico di Torino, PAOLA CARBONE, University of Manchester, DANIELE MARCHISIO, Politecnico di Torino, OMAR MATAR, Imperial College London — Polymer nanoparticles (NP) formation is investigated by a multi-scale modelling approach focusing on poly-e-caprolactone (PCL) self-assembly in acetone-water mixtures via flash nano-precipitation (FNP). The control of the final NP size at the outlet of the mixer and the evolution of the particle size distribution, cluster mass distribution (CMD), are guaranteed by a suitable population balance model (PBM) closed with the quadrature method of moments (QMOM). The CMD moments are transported and coupled with computational fluid dynamics (CFD). NP size is characterised in terms of mean radius of gyration, expressed through Flory theory. Molecules are treated as building blocks undergoing aggregation once the solubility limit is exceeded. The rate at which two NP collide and aggregate is expressed by the aggregation kernels, built upon molecular dynamics simulations. Turbulent fluctuations on NP formation are taken into account thanks to the direct quadrature method of moments coupled with the interaction-and-exchange-with-the-mean (DQMOM-IEM) method. Favre-averaged continuity and Navier-Stokes equations are implemented, in order to consider the density fluctuations of the system. The model is validated against experiments, showing an excellent agreement.

1Computational resources were provided by HPC@POLITO, at Politecnico di Torino.

Sunday, November 24, 2019 3:48PM - 5:19PM – Session G03 Focus Session: Fish Swimming Kinematics and Hydrodynamics I

3:48PM G03.00001 Fish out of water: hydrodynamics from swimming to jumping1, ALEXANDRA H. TECHET, Massachusetts Institute of Technology — Aquatic jumping is widely revered with awe by humans, as evidenced in the popularity of the Discovery Channels “Shark Week” episodes on jumping sharks. Jumping by any organism requires high bursts of power and muscular coordination. Aquatic jumpers in particular must produce thrust in manners compatible with the transition from one fluid media to the next, specifically accounting for the drastic drop in fluid density (and thus force-producing ability) between water and air. Jumping for food requires more precision than jumping for migration or escape. Organisms ranging in size from large marine mammals and sharks (length O(10m)) to small copepods (length O(0.01m)) have developed aquatic jumping strategies compatible with their size and survival goals (e.g., prey capture, escape, mating or migration). Larger animals tend to employ burst swimming for short periods before water exit, and smaller swimming fish tend to use S- or C-start acceleration maneuvers to generate jump thrust. Even smaller organisms such as copepods exhibit jumping behaviors using unique hydrodynamic mechanisms for impulsive vortical formation. However, water escape by engineered machines has yet to be mastered. Water-to-air transitions in vehicles are typically accomplished through high-momentum, high fuel consumption, artificial jets or propellers. The synergistic multi-propulsor relationships identified from fish swimming investigations could be paradigm-shifting for the design of biotinspired multiphase vehicles. This talk will discuss how studies of freely swimming fish and flapping foils can inform jumping analysis and our understanding of fin-fin interactions in complex fish maneuvers.

1Acknowledgements to Profs. Leah Mendelson and Fotis Sotiropoulos for their invaluable collaborations on archer fish jumping hydrodynamics research. Funding provided by NSF CBET 1700978.
as well as in swimming at various speeds.

We quantitatively investigated the burst-and-coast gaits that minimize CoT in transport with and without target distance, less than that during undulatory propulsion at the same speed, properly switching between undulatory propulsion and coasting may reduce the cost of transport. We investigated the burst-and-coast gaits that minimize CoT during cyclic swimming. The derived optimal strategies for various types of swimmers all suggest that fish should change tail-beat frequency to control speed with a nearly constant tail-beat amplitude. Because drag during coasting is much less than that during undulatory propulsion at the same speed, properly switching between undulatory propulsion and coasting may reduce the cost of transport. We quantitatively investigated the burst-and-coast gaits that minimize CoT in transport with and without target distance, as well as in swimming at various speeds.

constructing a simulation-based performance map in the frequency-amplitude parameter space, we obtained the speed-specific optimal strategy that minimizes the cost of transport (CoT) during cyclic swimming. The derived optimal strategies for various types of swimmers all suggest that the drag magnitude is strongly influenced by the undulatory kinematics, much exceeding that of a rigid fish gliding at the same speed. By swimming fish, and found that the scaling trend of drag during undulatory swimming follows that of a rigid three-dimensional object, while Paris Diderot 7, France — Fish undulatory kinematics is not only a means to overcome drag, but also a source of it. By utilizing a computational approach that couples the Navier-Stokes equations with the equations of undulating body motion, we decomposed thrust and drag in swimming fish, and found that the scaling trend of drag during undulatory swimming follows that of a rigid three-dimensional object, while the drag magnitude is strongly influenced by the undulatory kinematics, much exceeding that of a rigid fish gliding at the same speed. By constructing a simulation-based performance map in the frequency-amplitude parameter space, we obtained the speed-specific optimal strategy that minimizes the cost of transport (CoT) during cyclic swimming. The derived optimal strategies for various types of swimmers all suggest that fish should change tail-beat frequency to control speed with a nearly constant tail-beat amplitude. Because drag during coasting is much less than that during undulatory propulsion at the same speed, properly switching between undulatory propulsion and coasting may reduce the cost of transport. We quantitatively investigated the burst-and-coast gaits that minimize CoT in transport with and without target distance, as well as in swimming at various speeds.

This work was supported by the Office of Naval Research under ONR Award No. N00014-17-1-2759. The authors also wish to thank the Syracuse Center of Excellence for Environmental and Energy Systems for providing funds used towards the purchase of lasers and related equipment.

Numerical analysis of the force generation mechanism in a stingray inspired circular plan-forms. Seth Brooks, Melissa Green, Syracuse University — A two-degree-of-freedom fish model was investigated to understand the phenomenological relationship between simplified fish body kinematics and wake vortex dynamics. Its design, construction, and actuation provide control over frequency, tail (posterior half of body) angle, caudal fin to tail relative angle, and phase offset between the two angles. The frequency and phase offset were fixed for all cases in the current work while the tail and caudal fin angles were varied to create eight cases. Phase-averaged velocity data was collected beside the posterior half of the model as well as in the wake of the model. Data was obtained using stereoscopic particle image velocimetry at multiple planes along the entire span of the caudal fin. It was found that the body-generated vortices did not significantly interact with the caudal fin. The caudal fin leading edge vortex detaches from the surface sooner in cases with larger maximum tail angle. The total circulation generated at the caudal fin trailing edge was found to be sensitive to trailing edge velocity while being relatively insensitive to freestream velocity. Finally, the shedding of vortices from the caudal fin trailing edge was found to usually, but not always, coincide with periods of trailing edge deceleration.

The ground effect on anguiliform swimming performance. Mohsen Daghooghi, University of Houston-Clear Lake, Uchenna Ogunka, Iman Borazjani, Texas A&M University — Sea lampreys are found in the northern and western Atlantic Ocean along shores of Europe and North America as well as in the shores of Great Lakes and nearby rivers. This species is anadromous; from their lake or sea habitats, they migrate up rivers to spawn. In other words, they specialized to efficiently swim not only in deep but also in shallow waters. Various studies have shown that certain types of fish swim close to a solid surface and when it is unconstrained or freely swimming in the cross-stream direction. It was found that the foil has stable equilibrium altitudes: the time-averaged lift is zero at certain altitudes and acts to return the foil to these equilibria. These stable equilibrium altitudes exist for both constrained and freely swimming foils and are independent of the initial conditions of the foil. In all cases, the equilibrium altitudes move farther from the ground when the Strouhal number is increased or the reduced frequency is decreased. Potential flow simulations predict the equilibrium altitudes to within 3-11%, indicating that the equilibrium altitudes are primarily due to inviscid mechanisms. In fact, it is determined that stable equilibrium altitudes arise from an interplay among three time-averaged forces: a negative jet deflection circulatory force, a positive quasi-static circulatory force and a negative added mass force. At equilibrium, the foil exhibits a deflected wake and experiences a thrust enhancement of 4-17% with no penalty in efficiency as compared to a pitching foil far from the ground.

The computational resources for this study were partly provided by the High-Performance Research Computing (HPRC) group at Texas A&M University.
5:06PM G03.00007 Fluid-mediated stable equilibria for two-dimensional schools\textsuperscript{1}. MELIKE KURT, KEITH MOORED, Lehigh University, MOORED LAB TEAM — Fish oscillate their fins and tails and create disturbances in the form of vortical wake structures. These fish often operate in groups or collectives, which alter these structures and consequently their force production. In the 1970s, Sir James Lighthill hypothesized that for fast locomotion, hydrodynamic interactions could give rise to orderly patterns in fish collectives without the need for collective decision-making or active control mechanisms. Previous studies tested this hypothesis by studying the force equilibria in one-dimensional schools where swimmers are arranged along the stream-wise direction in in-line configurations. Here, our goal is to investigate the existence of these equilibrium points in two-dimensional schools by varying synchrony, as well as the cross-stream and stream-wise spacings between two hydrofoils. To this end, flow field analysis, as well as force measurements, will be conducted to draw a relation between the fluid-mediated forces and vortical interactions within the flow-field.

\textsuperscript{1}NSF Career Award 1653181

Sunday, November 24, 2019 3:48PM - 5:06PM — Session G04 Suspensions: General III 203 - Alexander Zinchenko, University of Colorado Boulder

3:48PM G04.00001 Tunable solidification of cornstarch under impact: how to make someone walking on cornstarch sink. RAN NIU, MEERA RAMASWAMY, Cornell University, CHRISTOPHER NESS, University of Cambridge, ABHISHEK SHETTY, Anton Paar USA, ITAI COHEN, Cornell University — Hundreds of Youtube videos with millions of views show people running on a mixture of cornstarch and water. These videos demonstrate a general phenomenon in fluid mechanics that dense shear thickening suspensions can solidify under impact. Such processes can be mimicked by impacting and pulling out a solid plate from the surface of a thickening cornstarch suspension. Here, using both experiments and simulations, we show that by applying fast oscillatory shear transverse to the primary impact or extension directions we can tune the degree of suspension solidification. The forces acting on the impacting surface can be modified by varying the dimensionless ratio of the orthogonal shear to the compression and extension flows. Simulations show that varying this parameter changes the number of particle contacts governing solidification. To demonstrate this strategy in an untethered context, we show that the sinking speed of a cylinder dropped onto the cornstarch suspension can be varied dramatically by changing this dimensionless ratio. These results suggest that applying orthogonal shear in the context of people running on cornstarch would de-solidify the suspension and cause them to sink.

4:01PM G04.00002 Dynamic Jamming Around a Cylinder Moving Through a Shear Thickening Suspension. OLAV RMCKE, NTNU, Department of Energy- and Process Engineering, IVO R. PETERS, University of Southampton, Faculty of Engineering and Physical Sciences, R. JASON HEARST, NTNU, Department of Energy- and Process Engineering — Dense suspensions exhibit a rich set of behavior, such as shear banding, rheochoa, continuous and discontinuous shear thickening, and in some extreme cases even jamming. Dynamic jamming by shear, which is the focus here, is caused by the formation of a frictional contact network between grains as stress increases. Previous works have shown, in separate experiments, that a transient jammed region can develop under pulling, pushing and shearing. How these different scenarios interact when they appear in a single system, however, is unknown. By dragging a cylinder through a suspension of cornstarch and water, we are able to track the shape of the jammed region as it propagates through the suspension using PIV on the free surface. This work makes it possible to directly compare the propagation of the jammed region, in a system where pulling, pushing and shearing coexist.

4:14PM G04.00003 Flow regimes of a neutrally buoyant suspension in the wake of a circular cylinder\textsuperscript{1}. RAPHAEL MAURIN, MATTHIEU MERCIER, LAURENT LACAZE, Institut de Mécanique des Fluides de Toulouse, Université de Toulouse, CNRS, JEFFREY MORRIS, Levich Institute and Department of Chemical Engineering, City College of the City University of New York — While the rheology of suspensions has been mainly studied in the Stokes regime, fluid mechanical applications such as blood flows or sediment transport at finite particle Reynolds number require a more general understanding. From this perspective, we revisit experimentally the well-known flow around a cylinder, considering a neutrally-buoyant suspension instead of a pure fluid. Varying the particle Reynolds number, \( Re \), and the solid volume fraction, \( \phi \), we investigate various wake structures, from the laminar creeping flow to the Karman vortex street. We characterize the corresponding stability map as a function of \( Re \) and \( \phi \). The presence of particles affects the shape of the wake of the cylinder in a non-trivial way, which cannot be accounted for by a modification of the effective fluid viscosity due to particle loading. Furthermore, the presence of the particles alters the critical values of the Reynolds number at which transitions in the flow regimes occur. These results can also be interpreted by the local behavior of the particles relative to the fluid near specific features of the flow around the cylinder.

\textsuperscript{1}NEMESIS Chair, IDEX UNITI

4:27PM G04.00004 Fast Lagrangian-averaged transport of particles in streaming flows\textsuperscript{1}. MATHIEU LE PROVOST, JEFF D. ELDREDGE, University of California, Los Angeles — Viscous streaming is an efficient method to transport, trap or cluster inertial particles in a fluid, used in biomecine, microfluidics... Nonetheless, current methods to simulate the long term behavior of inertial particles are computationally expensive due to the nonlinearity and stiffness of the Maxey-Riley equation when applied to the wide disparity of time scales of oscillation and mean convection in streaming flows. To handle these issues, we propose a novel framework called Fast Lagrangian Averaged Transport to efficiently compute the Lagrangian-averaged motion of inertial particles. Two key ingredients are used to get this improved performance. First, we derive an asymptotic expansion for an Eulerian inertial particle velocity field for small Stokes number about the Eulerian fluid particle velocity. Secondly, we decompose the motion of an inertial particle into a mean (slow) and fluctuating (rapid) component derived from an Eulerian disturbance field evaluated at the mean Lagrangian position of the particle. Linearized equations for the disturbance field are derived, and solved using an Immersed Boundary Method. Our method is assessed on the transport generated by one or two weakly oscillating cylinders. Computations are up to 300 times faster with this new method.

\textsuperscript{1}National Science Foundation under grant 1538824
at high Karlovitz numbers. HSU CHEW LEE, PENG DAI, Southern University of Science and Technology, ZHENG CHEN, and is found to be nearly independent of the Reynolds number. Our results are compared with others in the literature.

In this study, we investigate the fractal nature of the surface of several turbulent spherical flames subject to decaying turbulence at varying Reynolds number. We find that the flame surface has a fractal dimension that varies in time, remaining close to 2.4, although the limited size of studies on interfaces in isothermal turbulence, it has been postulated that turbulent flame surfaces exhibit scale-invariant or fractal properties. The flame surface is stretched, folded, and wrinkled on a multitude of length scales ranging from the Kolmogorov scale to the integral scale. Consistently with this study, we investigate the front propagation and burning characteristics in the MILD mode through high-fidelity direct numerical simulation (DNS). As a baseline condition, a lean hydrogen-oxygen mixture at an equivalence ratio of 0.7 is considered, which is diluted with nitrogen.

The turbulence condition falls under the thin reaction zone and high Lewis number. The resulting normalized TKE transport equation involves only a small set of parameters. The Lewis behavior of the terms in the TKE transport equation is analyzed and scaling terms proposed for the thin reaction zone is examined for a broken plane, providing temperature-resolved results and allowing for the study of dispersion as particles travel through the flame. Our results are compared to non-reacting turbulent flows to determine what differences exist between high Karlovitz number flames and non-reacting turbulence. Particular attention is given to recovering Richardson’s scaling and determining how the scaling changes through the flame.

Measurement (Re = 10^4) of a one dimensional lattice of discs in a quasi-two-dimensional geometry with the trajectory of the centres of the disks lying in a plane. We induce initial positional perturbations over a configuration in which the disks are uniformly spaced with their separation vectors and normals aligned, and perpendicular to gravity. For various perturbation wavenumbers and interparticle separations, we find two classes of behaviour: (i) a transient wave of orientations coupled with number-density fluctuations and (ii) a clumping instability resembling that of spheres. J.M. Crowley, J. Fluid. Mech. 45, 151 (1971), decorated with orientations. We construct the equations of motion for displacements and orientations using pairwise addition of forces and torques. R. Chajwa et. al. PRL 122, 224501 (2019). Linear stability analysis demarcates a phase boundary between neutrally stable and unstable regimes in the plane of wavenumber and lattice spacing, consistent with our experiments. We predict non-modal growth in this plane, with a critical density of the lattice below which all wavenumbers are asymptotically stable, showing that orientable particles need not be subject to the inevitable clumping instability of spheres.

NSF Award 1805921

1 Research reported in this publication was supported by the King Abdullah University of Science and Technology (KAUST).

Session G05 Turbulent Flames 204 - Peter Hamlington, University of Colorado Boulder
using POD based Finite Time Lyapunov Exponents (FTLEs). SINA RAFATI, University of Texas At Austin, NOEL CLEMENS, University of Texas At Austin — Rare events are defined as the excursions of a dynamical system toward unwanted conditions with possible catastrophic consequences. To that end, the focus of this study is to investigate the interaction of turbulence with a jet flame to better understand the occurrence of rare events in combustion such as flashback, extinction or blowout. Kraichnan (1965) has shown that there is a strong correlation between the existence of rare events and a fluid’s memory. As a consequence, the persistence of an initiated perturbation in a dynamical system for time-scales comparable to the large-scale flow time-scales might lead to rare events. In this study, 20 kHz Particle Image Velocimetry (PIV) has been used for velocity measurement of lifted methane-air turbulent flames. Two Coherent Evolution-90 diode-pumped Nd:YLF lasers were used to create 527nm pulses for PIV. The proper orthogonal decomposition (POD) method was utilized to obtain a compact representation of the velocity field. Then, the high-dimensional velocity field was projected into a lower-dimensional space for Lower Order Reconstruction (LOR) of the flow field with the aim of bringing new insight to the contribution of various scales in the chaotic development of the flow. Finally, Lagrangian coherent structures (LCSs) are obtained as ridges of FTLE maps to study the flow topologies as a function of space and time. Our results are representing how LCSs interact with flame as they are approaching to the flame front.

---

**4:53PM G05.00006 Compressive Dynamics of Fast Turbulent Flames**

RACHEL HYTIVOVC, JONATHAN SOSA, JESSICA CHAMBERS, KAREEM AHMED, ALEXEI POLUDNENKO, VADIM GAMEZO, None — h — abstract —

The research characterizes the dynamics of compressible flame-turbulence interactions for propagating fast flames. A Turbulent Shock Tube with a series of turbulence inducing plates has a large viewing area to capture the flame dynamics with various optical diagnostics, including high-speed PIV and schlieren. The experimental results show that the turbulent Mach number, $M_f$, within the flame increases non-linearly relative to the flame propagation Mach number, $M_f$, and grows quickly for flames propagating faster than Chapman-Jouguet deflagrations ($M_f > 1$). This relationship shows that turbulence is self-generated by fast turbulent flames. Furthermore, the flames with $M_f > 1$ are intrinsically unsteady. They tend to accelerate and generate shocks. This acceleration is accompanied by the fast increase of $M_f$ and continues until shocks become strong enough to ignite a detonation. Slower flames with $M_f < 1$ show, little or no self-generated turbulence, and do not produce shocks. The results are highly relevant for hypersonic scramjet propulsion engines and compressible shock-laden turbulent reacting flows in rotating detonation engines.\[2/-abstract-

---

1This work is sponsored by the Air Force Office of Scientific Research (FA9550-16-I-0403, Program Manager Dr. Chiping Li) and the American Chemical Society Petroleum Research Fund (S7453-1D19).

**5:06PM G05.00007 Kinetic Energy Backscatter in High-Speed, Compressible Reacting Turbulence**

ARNAB MOITRO, ASHWATH SETHU VENKATARAMAN, ALEXEI POLUDNENKO, Texas A&M University — Previous studies have shown that in certain reacting flow regimes, near the flame region, the direction of kinetic energy cascade reverses compared to the non-reacting turbulence, and is primarily directed from small scales to large scales on average. Studying this phenomenon (often termed backscatter) is important for developing large-eddy simulation (LES) models for turbulent combustion. Previous studies, however, were limited to relatively low-Mach number flows in idealized geometries. In the present work, we study the backscatter in highly-compressible regimes characterized by large Reynolds numbers. In particular, we present direct numerical simulations (DNS) of the flow in a Turbulent Shock Tube facility designed at the University of Central Florida with the goal of probing the dynamics of turbulent flames in such fast regimes. We quantify the backscatter by low-pass filtering the primitive variables in the DNS at various scales, and evaluating the the sub-filter scale terms in the equation for the transport of kinetic energy. Finally, we discuss the implications of these results for the development of the new generation of LES models for high-speed, compressible reacting flows.

**5:19PM G05.00008 Direct Numerical Simulation of Flame-Wall Interaction in a Constant Volume Vessel with a Crevice**

YUKI MINAMOTO, Tokyo Institute of Technology, ANDREA GRUBER, SINTEF Energy Research, MAMORU TANAHASHI, Tokyo Institute of Technology — Understanding of flame-wall interaction phenomena is important for further reduction of pollutant formation and enhancement of efficiency of various combustors. Many combustion devices involve crevice regions which yields much smaller length scale than the size of a combustor. In such crevice regions, the wall heat loss and turbulent mixing could be modified due to the geometrical effect of the crevice, resulting in more complex flame-wall interaction, which is not fully understood. In this study, a DNS of turbulent methane-air flame-wall interaction in a constant volume vessel with a crevice has been performed to understand the combustion physics near the crevice region. The present DNS configuration is chosen based on a typical IC engine combustion in terms of domain and crevice sizes and turbulent combustion condition on the combustion diagram, although methane-air combustion at atmospheric pressure was considered. The visual examination of DNS results shows that the mixture in the main domain is entrained and mixed with the mixture in the crevice. On the other hand, the mixture in the crevice is cooled as thermal boundary layer develops. The DNS results also revealed several interesting thermochemical and fluid dynamic aspects relevant to near-wall crevice combustion.

---


**3:48PM G06.00001 Modelling peak supersonic heated jet noise at fixed jet or fixed acoustic Mach number using an acoustic analogy-based non-parallel flow asymptotic approach**

MOHAMMED AFSAF, University of Strathclyde, ADRIAN SESCUC, Mississippi State University, EDMONDO MINISCI, University of Strathclyde — This presentation provides a summary of our recent results on supersonic jet noise prediction using the generalized acoustic analogy approach. The analogy poses a formula for the far-field sound as a convolution product of propagator tensor and the fluctuating Reynolds stress auto-covariance tensor. We found that approximating the propagator tensor using a non-parallel flow asymptotic approximation (AfSar et al., 2019. Phil. Trans. A, in press) of the vector Green’s function that satisfies the adjoint linearized Euler equations allows accurate prediction of the peak jet noise. That is, in the small observation angle region, the predictions remain within 1-2 dB of experimental data up to a Strouhal number (based on jet diameter) of at least 0.5. We base the analogy on a model of the Reynolds stress auto-covariance that agrees with Large-Eddy Simulations in isothermal conditions. In general, our predictions recover both the spectral quietening observed in heated jets at fixed acoustic Mach number (AfSar et al. AIAA J., vol.49, p.2522, 2011) and the noise enhancement at fixed jet Mach number. We observed that in either of these cases, using the simpler non-spreading locally parallel flow based Greens function results in a significant under prediction of sound.
4:01PM G06.00002 Large eddy simulations of supersonic twin rectangular jets including screech\textsuperscript{1}. JINAH JEUN, GAO JUN WU, SANJIVA K LELE, Stanford University — In this work the aeroacoustics of supersonic jets including screech tones, issuing from military-type twin rectangular nozzles with aspect ratio of 2.1 and design Mach number of 1.5, is studied numerically. Large eddy simulations (LES) are performed using the unstructured compressible flow solver ChalLES developed by Cascade Technologies, to fully replicate the nozzle geometry including the upstream duct with flow divider and sharply turning converging-diverging nozzle. Equilibrium wall-model is used to account for the internal boundary layers. Three different nozzle operating conditions, which correspond to two over-expanded conditions and one ideally-expanded condition, are considered. At each operating condition, the far-field acoustic spectra are obtained from the near-field LES data and the far-field flow statistics and the far-field acoustic modes are compared with experiments conducted at the University of Cincinnati, to examine predictive capabilities of the LES solver. Interactions between closely placed two jet plumes as they spared are captured, and the effects of nozzle operating conditions on them are discussed.

\textsuperscript{1}This work was supported by the Office of Naval Research under Grant No. N00014-18-1-2391.

4:14PM G06.00003 Sound Sources in Subsonic Twin Turbulent Jets, ARNAB SAMANTA, NISHANTH MUTHICHUR, SANTOSH HEMCHANDRA, Indian Institute of Science — We analyse the sound from merging identical twin jet plumes. The far-field acoustic spectra are obtained using the near-field LES data and the far-field flow statistics and the far-field acoustic modes are compared with experiments conducted at the University of Cincinnati, to examine predictive capabilities of the LES solver. Interactions between closely placed two jet plumes as they spared are captured, and the effects of nozzle operating conditions on them are discussed.

4:27PM G06.00004 Wavenumber-frequency spectra of wall-pressure fluctuations in compressible turbulent channel flow\textsuperscript{1}. YI LIU, KAN WANG, MENG WANG, University of Notre Dame — Knowledge of wall pressure fluctuations in wall-bounded flows is important for predictions of structural vibration and noise. In this study, direct numerical simulations with a high-order, non-dissipative finite difference scheme are employed to accurately capture the spatiotemporal characteristics of wall pressure fluctuations in compressible turbulent channel flows at Mach 0.4 and friction Reynolds number of 180. Acoustic peaks are barely visible in the one-dimensional streamwise wavenumber-frequency spectra. However, in the two-dimensional wavenumber-frequency spectra at the zero spanwise wavenumber, acoustic peaks associated with both longitudinal and oblique propagating waves are clearly identifiable, although they are several orders of magnitude weaker than the convective peak. The number of oblique-wave peaks at each frequency matches the theoretical prediction for duct acoustic modes. The effect of a small two-dimensional hump on the channel surface is also investigated. The results suggest that acoustic scattering by the hump vastly increases the acoustic energy for both longitudinal and oblique modes.

\textsuperscript{1}Supported in part by ONR Grant N00014-17-1-2686

4:40PM G06.00005 Analysis of Screech Generation in a Cold Supersonic Rectangular Jet with Large-Eddy Simulations\textsuperscript{1}. GAO JUN WU, Department of Aeronautics and Astronautics, Stanford University, SANJIVA LELE, Department of Aeronautics and Astronautics and Department of Mechanical Engineering, Stanford University, JINAH JEUN, Center for Turbulence Research, Stanford University — Screech is an aeroacoustic phenomenon found in non-ideally expanded supersonic jets. Using large-eddy simulations, this work studies screech generation for a cold under-expanded rectangular jet at several Nozzle Pressure Ratios (NPRs) around maximum screech. The aspect ratio of the converging-diverging rectangular nozzle is 4:1 and the design Mach number is 1.44. In the noise spectra, screech tones are observed at near-field locations. Spectral Proper Orthogonal Decomposition (SPOD) techniques are used to extract the temporal-spatial coherent structures from the LES data. At the screech frequency, a flapping motion is detected along the minor-axis plane, and a spatially modulating antisymmetric standing wave pattern is observed in the near field as a result of oppositely travelling hydrodynamic and acoustic wave packets. In contrast to the conventional belief that screech feedback is closed by upstream-travelling acoustic waves outside the jet, recent research has indicated that the closure mechanism could arise from an internal acoustic mode supported by jet plumes. To further investigate this hypothesis, a linear instability analysis for the jet flow will be conducted, and the results will be compared with the SPOD data.

\textsuperscript{1}This work is supported by ONR Grant No. N00014-18-1-2391. Computational resources are provided by the INCITE and XSEDE programs.

4:53PM G06.00006 Linear stability analysis of supersonic jet screech. MICHAEL KARP, TIM FLINT, M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University — Jet screech is an undesirable flow phenomenon which can pose severe limitations on the operation of jet engines. Screech is commonly understood as a feedback cycle involving interactions between instability waves, shear layers, shocks and acoustic waves. Our study seeks to advance the insight into the physics of jet screech by means of global linear stability theory. We consider steady laminar axisymmetric jets at supersonic conditions. Both under-expanded and perfectly expanded jets are investigated at various exit Mach numbers. The analysis connects the occurrence of screech to an absolute instability of the under-expanded jet. The features of the eigenmodes are discussed and their relevance to the elements of the screech cycle is explored. An adjoint analysis quantifies the receptivity of jet screech to internal and external perturbations.

5:06PM G06.00007 A New Instance of Crackle Noise. JOSEPH MATHEW, SUMIT PATEL, Indian Institute of Science — Intense crackle noise has been identified and studied in high temperature, supersonic jets of engines operating at high specific thrust. We observed a new instance of crackle noise in cold under-expanded supersonic jets (Mach 1.5, Reynolds number 100,000) impinging on plane wedges. LES is by explicit filtering, with a new adaptive filtering method for shock capturing. Crackle fronts appear when a detached normal shock stands about half a jet-width from the wedge tip, and the jet column is about 4 jet-widths (nozzle to wedge tip). The edges of these oscillating shock interact with the jet’s bounding shear layer structures to emit acoustic wave fronts that steepen as they travel into the far field. Pressure signals exhibit sudden rises followed by gentle relaxations, intermittently. Consistently, pressure distributions along lines in the ambient show the existence and arrival of a sequence of sharp compression fronts. Skewness exceeds 0.4 in the far field. Crackle fronts do not appear in other configurations such as a short column (steady shock), short column and thin wedge (steady, attached, oblique shock), and a long column, thin wedge (large amplitude column oscillations).
Approach provides a 3D map of the residence time, while the latter estimates the order of magnitude. Circulation in a plane normal to the axis of the vortex dominating the aneurysm sac is used to estimate the eddy turnover time. The former approach provides a 3D map of the residence time, while the latter estimates the order of magnitude. In the Eulerian approach, risk of rupture in untreated aneurysms and risk of thrombo-embolism in treated aneurysms. In this study, we use in vitro 4D MRI velocimetry to model blood flow in aneurysms.

Additional clinical observations indicate thrombus formation following aneurysm endovascular treatment perhaps due to inadequate blood flow. It is well known that blood flow residence times of critical stenoses significantly influence the formation of thrombosis and the risk of rupture. Blood flow residence times are not fully understood, and there is emerging evidence that blood flow residence times are associated with the formation of thrombosis in cerebral aneurysms.

The carotid phantom was constructed with a 3D printer based on the CT image of patient and the actual pulsatile flow. To reproduce the patient's blood flow measured by US Doppler probe. The blood mimicking glycerin aqueous solution was used as a working fluid. The overall experimental setup was designed to reproduce the patient's blood flow measured by US Doppler probe. The blood mimicking glycerin aqueous solution was used as a working fluid.

The carotid phantom was constructed with a 3D printer based on the CT image of patient and the actual pulsatile flow. To synchronize the pulsatile flow, a digital pulse was transmitted to the MR scanner as trigger when the wave form started. As a result, a time-resolved 3D pulsatile flow with a high spatiotemporal resolution (350 um iso and 25 ms) was obtained.

Assessment of cardiac output effects on bioprosthetic pulmonary valve behavior using magnetic resonance velocimetry. The geometry-independency of the estimation of turbulence production and corresponding irreversible pressure drop was demonstrated and quantified. The flow configurations are investigated where scattering of boundary layer instabilities lead to tonal noise generation including equidistant secondary tones. Despite the geometric symmetry at zero deg. incidence, the flows become non-symmetric with a separation bubble on one side of the airfoil. A separation bubble is also observed for the non-zero incidence flow. It is shown that the frequency motion of the separation bubbles induces a frequency modulation of flow instabilities developed along the airfoil boundary layer. The airfoil is at zero deg. angle of attack, and the flow quantities result in a complex vortex interaction mechanism at the trailing edge. Both amplitude and frequency modulations directly affect the velocity and pressure fluctuations that are scattered at the trailing edge, what leads to secondary tones in the acoustic radiation.

Assessment of cardiac output effects on bioprosthetic pulmonary valve behavior using magnetic resonance velocimetry. The geometry-independency of the estimation of turbulence production and corresponding irreversible pressure drop was demonstrated and quantified. The flow configurations are investigated where scattering of boundary layer instabilities lead to tonal noise generation including equidistant secondary tones. Despite the geometric symmetry at zero deg. incidence, the flows become non-symmetric with a separation bubble on one side of the airfoil. A separation bubble is also observed for the non-zero incidence flow. It is shown that the frequency motion of the separation bubbles induces a frequency modulation of flow instabilities developed along the airfoil boundary layer. The airfoil is at zero deg. angle of attack, and the flow quantities result in a complex vortex interaction mechanism at the trailing edge. Both amplitude and frequency modulations directly affect the velocity and pressure fluctuations that are scattered at the trailing edge, what leads to secondary tones in the acoustic radiation.

Funded by the American Heart Association and Stanford Child Health Research Institute

Assessment of Reynolds stress using 4D Flow MRI for estimating the irreversible pressure drop across a stenosis. This study demonstrates novel geometry-independency of the irreversible pressure drop across stenoses by quantifying the amount of turbulence production. Based on 4D Flow MRI-based assessment of turbulence mapping using a six-directional icosahedral (ICOSA6) flow encoding scheme for measuring the complete Reynolds stress tensor, the feasibility of 4D Flow MRI for the quantification of irreversible pressure loss was investigated in a range of voxel sizes and signal-to-noise ratios (SNR) by simulating 4D Flow MRI based on data from computational fluid dynamics (CFD). The geometry-independency of the estimation of turbulence production and corresponding irreversible pressure drop was investigated using several stenoses. Finally, experimental acquisitions using 4D Flow MRI with ICOSA6 flow encoding were used to demonstrate the assessment of the irreversible pressure drop.

Assessment of Reynolds stress using 4D Flow MRI for estimating the irreversible pressure drop across a stenosis. This study demonstrates novel geometry-independency of the irreversible pressure drop across stenoses by quantifying the amount of turbulence production. Based on 4D Flow MRI-based assessment of turbulence mapping using a six-directional icosahedral (ICOSA6) flow encoding scheme for measuring the complete Reynolds stress tensor, the feasibility of 4D Flow MRI for the quantification of irreversible pressure loss was investigated in a range of voxel sizes and signal-to-noise ratios (SNR) by simulating 4D Flow MRI based on data from computational fluid dynamics (CFD). The geometry-independency of the estimation of turbulence production and corresponding irreversible pressure drop was investigated using several stenoses. Finally, experimental acquisitions using 4D Flow MRI with ICOSA6 flow encoding were used to demonstrate the assessment of the irreversible pressure drop.

MRI-based flow residence time in intracranial aneurysms. The mechanisms behind development of thrombosis in cerebral aneurysms is not fully understood, and there has been emerging evidence that blood flow residence time is correlated/associated with the lesion. In addition, there have been clinical observations indicating thrombus formation following aneurysm endovascular treatment perhaps due to altered hemodynamics. Therefore, a characterization of the residence time can play an important role in the development of a biomarker for the risk of rupture in untreated aneurysms and risk of thrombo-embolism in treated aneurysms. In this study, we use in vitro 4D MRI velocimetry for several aneurysms performed in a 3 Tesla magnet at submillimeter resolution to compute the flow residence time using both Lagrangian and Eulerian frameworks. As per our previous study, by integration of the volumetric velocity field, Lagrangian tracks of massless tracers are computed, followed by the residence time calculation using the presence of fluid parcels in a local and global sense. In the Eulerian approach, circulation in a plane normal to the axis of the vortex dominating the aneurysm sac is used to estimate the eddy turnover time. The former approach provides a 3D map of the residence time, while the latter estimates the order of magnitude.
Remarkably, the extracted droplet is automatically and firmly trapped in the sound field which acts as an acoustic tweezer. High speed video was taken to display the dynamics of the interface during the procedure when the interface was broken deforming it against the incident direction of the sound beam. The direction of this force depends on the relative sound speeds and densities of the two fluids. We demonstrate the extreme case where a properly designed ultrasonic beam is used to break the interface and extract a single droplet from an interface of water to CCl$_4$. The experiment was repeated with a range of ultrasonic frequencies to study the effect on the droplet extraction process.

Ultrasonic extraction and trapping of a droplet from fluid-fluid interfaces.

Robert Lirette, Likun Zhang, Joel Mobley, The University of Mississippi — Ultrasonic waves provide a powerful means to exert forces on fluid objects (droplets and fluid interfaces) for the development of non-contact manipulation techniques or acoustic tweezers. Under proper conditions, a high-frequency sound wave can exert a pulling force on the interface of two immiscible fluids causing it to deform against the incident direction of the sound beam. The directional control of this force depends on several factors including the sound speed and density of the two fluids. This study demonstrates the feasibility and potential applications of this technique for the non-invasive manipulation of fluid interfaces.
4:01PM G08.00002 Capillary surfers: Self-propelling particles at an oscillating fluid interface1, GIUSEPPE PUCCI. Brown University and Institute of Physics of Rennes, DANIEL HARRIS, Brown University — In the present work, we explore the dynamics of millimeter bodies trapped at the air-water interface of an oscillating bath. The relative vertical motion of the body and the free surface leads to the generation of propagating capillary waves. We demonstrate that when the rotational symmetry of an individual particle is broken, the particle can steadily self-propel along the interface. Such self-propelled particles interact with one another through their mutual capillary wavefield and resultant fluid flows, and exhibit a rich set of collective modes characterized by a discrete number of equilibrium spacings for a given set of experimental parameters. Our results open the door to further investigations of this novel active system at the fluid interface. Ongoing work and future directions will be discussed.

1G. Pucci thanks the program CNRS Momentum for its support.

4:14PM G08.00003 Collective dynamics of dense particles at a liquid interface, ANTOINE LAGARDE, Institut Jean Le Rond D’Alembert, Sorbonne Universite, CHRISTOPHE JOSSERAND, LadHyX, Ecole Polytechnique, SUZIE PROTIERE, Institut Jean Le Rond D’Alembert, Sorbonne Universite — When two millimeter-sized identical objects are deposited at a liquid interface, the individual deformations they create may overlap, leading to an attractive capillary force. This everyday phenomenon that one can observe at the surface of water is central in many applications, from industrial processes where it is used to manufacture objects with a specific microstructure to Nature where fire ants gather into a raft to survive floods. The interaction between two identical spherical particles is now rather well understood, but the many-body interaction of non-identical aggregates still lacks a convincing description. Here, we experimentally study the aggregation of randomly distributed dense millimeter-sized beads placed at an oil-water interface. The particles are attracted by their neighbors and form an asymmetric mound called a granular raft. The interfacial deformation created by such an object exceeds by at least one order of magnitude the deformation of a single bead, leading to high capillary forces that strongly depend on the number of particles in each raft. Thanks to a precise understanding of the interaction between two rafts, we undertake a statistical description of the aggregation process of a n-body system into a single giant granular raft.

4:27PM G08.00004 Low-G inertial-capillary meniscus motions in a channel1, JOSH MCCCRANEY, PAUL STEEN, Cornell University, JOSHUA BOSTWICK, Clemson University — In low-g environments, residual accelerations can induce fluid reservoir reconfiguration, resulting in dynamic instabilities. Since channel geometries are prevalent aboard critical spacecraft fluid systems, such as propellants, cryogens, and wastes, understanding flow stability is crucial to ensure fluid is positioned and available when needed. In this work we analyze the normal oscillations of a low-g fluid in a rectangular channel, reporting fundamental frequencies for pinned, natural, and mobile contact line conditions.

1NASA Grant NNH17ZTT001N- 17PSI D

4:40PM G08.00005 Capillary Sorng of Particles by Dip Coating , BRIAN DINCAU, University of California Santa Barbara, MARTIN BAZANT, Massachusetts Institute of Technology, EMILIE DRESSAIRE, ALBAN SAURET, University of California Santa Barbara — High-throughput sorting or filtration of suspensions is a critical step in many industrial, geophysical, and biomedical processes. Here, we present a new scalable size-based separation technique which utilizes a dip coating meniscus as a tunable filter. When a plate is withdrawn from a liquid bath, a thin layer of liquid coats its surface. Below a given film thickness on the plate, the meniscus generates a strong capillary force at the stagnation point and prevents large particles from being entrained in the liquid film. We leverage the capillary filtration effect induced by the meniscus to sort particles by size. Indeed, the size threshold depends on the withdraw speed and fluid properties, so smaller particles are entrained while larger particles remain in the bath. We demonstrate this technique with bidisperse suspensions and explain how it could be applied to polydispersed suspensions or extended to biological suspensions. We rationalize our results in terms of dimensionless numbers (capillary and Bond numbers) and estimate the range of capillary number to separate particles of given sizes. This technique is well-suited for high-throughput operation due to the demonstrated scalability of industrial dip coating, combined with the clog-free nature of this meniscus-based filter.

4:53PM G08.00006 Steady Buckling of Viscous Capillary Jets , NEIL RIBE, CNRS, University Paris-Saclay, Orsay, France, JINGXUAN TIAN, XIAOXIAO WU, ANDERSON SHUM, Dept. of Mechanical Engineering, University of Hong Kong — Steady buckling (coiling) of thin falling liquid jets plays an important role in instability-assisted high-resolution printing and ultra-fine liquid dispensing, situations in which the effects of surface tension are likely to be significant. To understand better these capillary effects, we have performed experiments with sub-millimetric jets and ultra-low flow rates (∼10 ml/h) together with numerical simulations and linear stability analysis to study a hitherto unexplored reconfiguration dominated by capillarity with negligible inertia. We find that the critical fall height $H_c$ for coiling onset decreases with increasing flow rate, a tendency that is opposite to the zero surface-tension case and that has been previously documented only for inertia-dominated coiling. The enhanced resistance to buckling afforded by surface tension increases $H_c$ by up to a factor of 10 relative to the surface tension-free case. A regime diagram in the space of capillary number and jet slenderness agrees closely with the prediction of the linear stability analysis, but differs significantly from the analogous diagram for unsteady buckling of a compressed liquid bridge constructed previously by other workers.

5:06PM G08.00007 The Dynamics of Curved Thin Films Under Soluto-Capillary Forces , XINGYI SHI, MARIANA RODRIGUEZ-HAKIM, GERALD FULLER, ERIC SHAQFEH, Stanford University — Interfacial film dynamics is ubiquitous and is interesting in the presence of bulk and/or interfacial heterogeneity. To control the stability of thin films over different substrates, we need a fundamental understanding of the physical forces that affect the film, particularly when the film is heterogeneous. In the present study, we examine the thin film dynamics of a binary mixture subject to evaporation and drainage atop both a glass dome and an air bubble. Experimentally, we observe the film thickness profiles via a custom-made dynamic fluid-film interferometer. In the parallel computer simulations, we develop a lubrication theory to compute the film thickness evolution under the effects of capillarity, gravity, Marangoni forces, and van der Waals interactions with the substrate. We find the dynamics are quite complex and subject to long-lived time dependent states thus soluto-capillary forces stabilize the film. Moreover, we find that stabilizing van der Waals forces are crucial to create the conditions for Marangoni regeneration for drainage over a solid substrate. For a draining, heterogeneous thin film over a bubble surface, we find sustained, nonlinear thickness oscillations are rather easily accessible.
The finite-time Lyapunov exponent (FTLE) was used to analyze the unsteady flow around a pitching NACA 0012 airfoil. Wake characteristics (i.e., vortex size, peak vorticity, vortex orientation, vortex convection speed) are investigated by tracking the airfoil. The airfoil is pitched about the quarter chord point with an amplitude of $4\alpha$ at reduced frequencies of 2.6-5.8 for $Re_c = 12000$. Pitching symmetries of 50/50, 60/40, and 70/30, where the symmetry is defined as the fraction of the cycle spent in the pitch down and pitch up motion. Wake characteristics (i.e., vortex size, peak vorticity, vortex orientation, vortex convection speed) are investigated by tracking the vortices in the wake over the first chord length of development. Mean thrust/drag are calculated using a control volume approach including fluctuating terms.

This work was supported by the Office of Naval Research under ONR Award No. N00014-14-1-0418.
4:40PM G09.00005 Effect of Chordwise Flexibility on Propulsive Performance of High-Inertia Oscillating Foils1. PETER OSHKAI, DYLAN IVerson, MOSTAfA RAHMIPOUR, WALTFRDeE LEe, University of Victoria, TAKAHIRO KIWATA, Kanazawa University — This work studies the effects of chordwise flexibility, inertia and kinematic parameters on propulsive performance of an oscillating foil. Three low-aspect-ratio foils with different flexibilities were undergoing pitch and heave motions in a uniform flow at the Reynolds number of 80000. Forces exerted on the foil were directly measured using a load cell and were used to calculate the thrust and efficiency values. The phase-averaged flow velocity and out-of-plane vorticity in the wake of the foil were obtained using particle image velocimetry. The circulation in the wake was related to the loading on the foil by calculating the moments of vorticity with respect to the pitching axis of the foil. The generated thrust values monotonically increased as a function of the Strouhal number for the considered range of pitching angles. The deformation of the foil resulted in an increased wake width, leading to larger amplitudes of the instantaneous loading on the foil and higher thrust coefficient compared to a reference case of a rigid foil.

1Natural Sciences and Engineering Research Council of Canada (NSERC)

4:53PM G09.00006 Vortex dynamics of a rapidly pitching elliptical airfoil exhibiting dynamic stall at low Reynolds numbers. JOHN HRYNUK, US Army Research Lab, MINGJUN WEI, Kansas State University — The flow features and effects of dynamic stall have been traditionally studied for rotorcraft and in small bio-flyers. In rotorcraft focused dynamic stall studies a sinusoidal pitching motion is commonly used, while a perching maneuver for small vehicles is better represented by a constant pitch-and-hold motion. For pitch-and-hold motions the interaction of Dynamic Stall Vortex (DSV) and Trailing Edge Vortex (TEV) appear to have a significant impact on vortex circulation and convection. To expand the understanding of the interaction of DSV, TEV, and basic vortex shedding phenomenon, an elliptical airfoil shape was pitched about mid-chord with a constant dimensionless pitch rate. Flow fields were captured using PIV at a Reynolds number of Re = 12,000. Results show that depending on the phase angle of the airfoil, the flow is dominated by the DSV, TEV, or traditional bluff body shedding. These regions of phase angle will be delineated and vortex tracking methods will be used to evaluate convection behavior of the trackable structures.

5:06PM G09.00007 Dynamic stall experiments on various pitching motion profiles of an airfoil at high Reynolds numbers. JANIK KIEFER, Technical University of Denmark, CLAUDIA BRUNNER, MARCUS HULTMARK, Princeton University, MARTIN O. L. HANSEN, Technical University of Denmark — The combination of high Reynolds numbers and unsteady flow conditions depicts a challenge in experimental wind tunnel studies. Unsteady airfoil aerodynamics are commonly described by the reduced frequency k = omega c/2U, where a range of 0 < k < 0.25 characterizes steady to highly unsteady flow conditions. The Reynolds number scales proportionally with the flow velocity U, whereas the reduced frequency scales inversely proportional. In regular wind tunnels, this leads to unrealistically high pitching frequencies in experimental attempts to achieve high Reynolds numbers simultaneously with high reduced frequencies. Instead, this study takes advantage of a high-pressure flow facility, in which the density of compressed air promotes high Reynolds numbers, while low velocities below 5 m/s allow for low pitching frequencies and large angle amplitudes. A NACA0021 airfoil was equipped with surface pressure sensors to investigate distributed pressures and integrated forces at Reynolds numbers between one and five million. The present study elucidates the differences of various motion profiles on airfoil performance in comparison to the commonly employed sinusoidal pitching motion in dynamic stall conditions.

5:19PM G09.00008 Dynamic Stall Experiments on a Sinusoidally Pitching Airfoil at High Reynolds Numbers1. CLAUDIA BRUNNER, Princeton University, JANIK KIEFER, Technical University of Denmark, Princeton University, MARTIN O. L. HANSEN, Technical University of Denmark, MARCUS HULTMARK, Princeton University — The phenomenon of dynamic stall results in a lift overshoot experienced by an airfoil when its angle of attack is rapidly increased beyond the static stall angle. This overshoot is followed by a sudden drop in the lift force, and large hysteresis as the airfoil returns to its initial angle. Dynamic stall is observed in a variety of applications including helicopters and wind turbines, where it produces rapidly fluctuating loads on the blades. It is also seen in many biological systems, but they often use it advantageously. At low and moderate Reynolds numbers and reduced frequencies, this phenomenon has been extensively investigated, but due to the experimental challenges at high Reynolds numbers, only few studies have been conducted in this regime. In the current study, a NACA 0021 airfoil was oscillated sinusoidally around the static stall angle. A highly pressurized, low-velocity, wind tunnel was used to achieve Reynolds numbers up to 5 - 106, based on the chord length, and reduced frequencies up to 0.5. Forces and moments on the airfoil, as well as pressure distributions around its surface were recorded. Effects of Reynolds number, reduced frequency, mean angle of attack, and amplitude on the development of dynamic stall will be presented.

1Funded by the National Science Foundation (CBET 1652583)


3:48PM G10.00001 Effect of sweep on the laminar separated flows over finite-aspect-ratio wings1. KAI ZHANG, KUNIHKO TAIRA, University of California, Los Angeles — Separated flows over finite-aspect-ratio wings at low Reynolds numbers can exhibit rich wake dynamics across a range of aspect ratio, angles of attack, and sweep angles. In this talk, we focus on the effect of sweep on the three-dimensional flow physics behind finite NACA0015 wings. The post-stall wake of an unswept finite wing features a prominent tip vortex at the free end and unsteady wake vortices near the midspan. For a swept wing, a steady wake region forms from the midspan and the unsteady shedding region repositions toward the tip. The sweep-induced spanwise flow counteracts the roll-up of the flow over the wing tip, hindering the formation of the tip vortex. Corresponding to the spanwise variation of the vortical structures, the sectional lift coefficient is largest at the midspan, and decreases towards the wing tip. At large sweep angles, the wake of finite wings exhibits streaks of steady streamwise vortices that are embedded in the spanwise-undulated vortex sheets. In this talk, we discuss findings from a large parametric study and offer comparisons with wake behind unswept wings.

1We acknowledge the support by AFOSR (grant number FA9550-17-1-0222).
4:01PM G10.00002 Effect of boundary conditions on 3-D separation over an airfoil. SHELBY HAYOSTEK, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Boundary conditions greatly affect the wake topology around a finite-span, low aspect ratio airfoil. Two sets of boundary conditions were imposed on the airfoil and their effect was explored in water tunnel experiments. First, the model was cantilevered from a wall where the thickness of the boundary layer, at the junction between the airfoil and the wall, was about half the airfoils span. The second model was with two free ends, where the model was held by a thin rod attached to the wall. In both cases, the airfoils had a NACA 0015 profile with an aspect ratio of 2, and were tested at a range of angles of attack and chord Reynolds numbers of 600 and 1,000. In addition, the airfoils were either unswept or 30 degrees swept back. Stereo particle image velocimetry and dye flow visualization were used to capture the flow field. It was found that for the unswept case, the boundary layer greatly affected the flow structures in the wake, with a dual vortex system forming at the tip. However, for the sweep case, the spanwise velocity component pushed the wake towards the tip and the effect of the boundary condition was smaller compared to the unswept cases.

4:14PM G10.00003 Three-dimensional separation over a finite span NACA 0015 airfoil. JACOB NEAL, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Three-dimensional separation over a finite span, cantilevered NACA 0015 airfoil is influenced by several parameters. The effects of aspect ratio, Reynolds number, and angle of attack were explored in a series of wind tunnel experiments. Oil flow visualization (OFV) was performed on wings of aspect ratio four, two, and one. For all aspect ratios, it was seen that the Reynolds number did not affect the structures in the oil. For angles of attack above stall, two distinct foci were formed, one near the root and another counter-rotating near the wing tip. In addition, for a couple of cases, the flow field over the airfoil and the structures in its wake were explored using stereo particle image velocimetry at Reynolds number 330,000 for aspect ratio of two. At the higher angle, a clear three-dimensional separation bubble was present at the middle of the wing, and a surface normal vortex was seen to emerge from the focus points seen in the OFV and bend downstream into the flow.

4:27PM G10.00004 Dynamic stall and its post-stall lift decay1. SABRINA HENNE, KAREN MULLENERS, EPFL — Dynamic stall occurs when an airfoil undergoes dynamic changes in the angle of attack beyond its static stall angle. It can be observed for example on helicopter or wind turbine blades. Dynamic stall is associated with an increase in maximum lift, but this initial peak is followed by strong load fluctuations. These fluctuations can cause vibrations, structural damage, and failure. To identify the origin of the decay of post-stall lift fluctuations, the flow around a flat plate undergoing a ramp-up motion from an angle of attack of zero to an angle beyond the static stall limit was investigated using time-resolved flow field and load measurements. Immediately following the ramp-up motion, multiple subsequently shed leading edge vortices are observed and their separation coincides with local maxima in lift. The magnitude of these local lift maxima decays in time. A detailed analysis of the size and trajectory of the dominant leading edge vortices using Eulerian and Lagrangian methods helped to characterise the decay of post-stall lift peaks and quantify the importance of variations in the development of the leading edge vortices and changes in the wake topology on the airfoil’s post-stall performance.

4:40PM G10.00005 Time-Resolved PIV Airwake Measurements of a Frigate Ship Model1. ZHENG ZHANG, DHUREE SETH, EBENEZER GNANAMANICKAM, GORDON LEISHMAN, Embry-Riddle Aeronautical University, Daytona Beach, FL — The full ship model with standard Navy SFS2 geometry was tested in a 6 ft (W) by 4 ft (H) by 12 ft (L), closed circuit, low-speed wind tunnel. A time-resolved planar PIV system was used to measure the temporal flow field at the centerline of the deck behind the funnel and hangar structures. The measurements were conducted at Reynolds numbers up to 8 million based on the length of the ship model. The flow field was noted to be dominated by two major structures, namely the wake of the funnel and the shear layer emanating from the superstructure of the ship and top of the hangar. The reattachment behind the hangar structure induced the shear layer reattach to the surface at approximately the middle of the deck, resulting in a highly energetic turbulent flow close to the surface. Dynamic model decomposition of the time-resolved flow field revealed existence of multiple dominant frequencies, indicating complex vortex flows in the vicinity of the deck. The ship model was also tested in a simulated atmospheric boundary layer, the results suggesting that the associated velocity gradient and higher turbulence weakens the shear layers produced over and near the deck.

4:53PM G10.00006 Effects of spanwise inhomogeneity on wake dynamics of a cylinder1. CHAO JIANG, YEFEI YANG, SHUJIN LAIMA, HUI LI, Harbin Institute of Technology — The flow over a twisted cylinder at low Reynolds numbers of Re=30-300 is investigated using direct numerical simulation based on the finite volume method. The elliptic cross-section of the cylinder is rotated along its axial direction, thus resulting in inhomogeneity of the geometry. For comparison, the flow past a smooth and a wavy cylinder is also calculated. The twisted cylinder achieves reductions of approximately 90% both in mean drag and lift fluctuation compared with smooth and wavy cylinders. When Re>160, the time trace of drag and lift for the twisted cylinder reveals the presence of multi-frequency oscillations, which is different from the other two. The plot of Strouhal number vs. Reynolds number for the twisted cylinder strikingly exhibits four discontinuities, while only two for the smooth cylinder. The first discontinuity corresponding to the start point for transition of the flow, moves towards a lower critical Reynolds number.

5:06PM G10.00007 Boundary Layer Separation from Sports Balls with Seams. ANDREW SMITH1, BARTON SMITH2. Utah State University — Results of a study on the behavior of boundary layer separation of various sports balls with seams using particle image velocimetry (PIV) is presented. Our primary focus is the boundary layer state at separation, the location of separation, and the angle formed by the separated shear layer. The study uses moving balls in free space rather than a wind tunnel, which may alter critical pressure gradients and affect separation behavior. The bulk of the results presented will be for non-rotating major league baseballs. The seams on baseballs play two distinct roles: 1) the may cause the laminar boundary layer to become turbulent (when located on the front of the ball) and 2) they often form the separation location (when on the back of the ball). The effect of surface roughness (scuffs) on boundary layer separation is also studied. To better understand the effects of seams, other seamless sports balls are examined including a cricket ball, a smooth ball, a sliotar (hurling ball), an artificially roughened leather baseball, and a printed baseball with exaggerated seams.
5:19PM G10.00008 Separating and reattaching turbulent boundary layer due to an unsteady adverse pressure gradient

JUNSHIN PARK, DONGHYUN YOO, Pohang University of Science and Technology — Turbulent boundary layer subject to an unsteady adverse pressure gradient (APG) is studied using direct numerical simulation (DNS). The unsteady APG is imposed with a suction/blowing velocity profile at the top numerical boundary to evoke boundary layer separation and reattachment. In particular, the suction/blowing velocity profile is varied sinusoidally in time at several different reduced frequencies: \( k = 0.75, 0.75 \) and \( 0.25 \). Results have shown that for reduced frequencies \( k = 0.75 \) and \( 0.25 \), flow features differ greatly from those of separated flow with a steady APG. The separated shear layer lifts up at the first half-period of the APG oscillation. Flow near the reattachment region is entrained back inside the separation bubble. Large vortices are subsequently generated and convect downstream while the separated shear layer leans down to the wall at the latter half-period of the APG oscillation. Differences in time-averaged turbulent kinetic energy are also observed compared with the steady counterpart. To investigate the origin of such differences between flows with steady and unsteady APG, dynamic mode decomposition is also applied. Identified dynamic modes show features related to the aforementioned unsteady behavior.

This work was supported by Samsung Research Funding Center of Samsung Electronics under Project Number SRFC-TB1703-01 and National Research Foundation of Korea (NRF) under Project Number NRF-2017R1E1A1A03070514.


3:48PM G11.00001 Molecular Tagging Velocimetry Study of High Reynolds Number Turbulent Pipe Flow in Cryogenic Helium, HAMID SANAVANDI, Department of Mechanical Engineering, Florida State University, Tallahassee, Florida, SHIRAN BAO, National High Magnetic Field Laboratory, Tallahassee, Florida, YANG ZHANG, Florida Center for Advanced Aero-Propulsion, Tallahassee, Florida, WEI GUO, National High Magnetic Field Laboratory, Tallahassee, Florida, LOUIS CAFFEESTA, Florida Center for Advanced Aero-Propulsion, Tallahassee, Florida — Cryogenic helium-4 has considerable potential in fluids research due to a very small kinematic viscosity, suitable to generate and study high Reynolds number turbulent flow within a compact laboratory apparatus. However, studying the flow in helium-4 has been challenging largely due to the lack of effective visualization and velocimetry techniques. Here we have assembled a novel instrumentation including a 335 cm long horizontal cryogenic helium channel with a square 4cm\(^2\) cross-section. This allows us to generate fully-developed turbulent pipe flows with Reynolds numbers above \( 10^6 \). We implement a unique molecular tagging velocimetry (MTV) method based on tracking two parallel thin He\(_2\) molecular tracer lines created perpendicular to the pipe wall with an adjustable distance via femtosecond laser-field ionization. By observing the displacement and distortion of the tracer lines, we can measure the near-wall mean velocity profile, velocity fluctuation profile, as well as both the transverse and longitudinal spatial velocity correlations. We also report the pressure drop data acquired from the experimental channel using a differential pressure transducer, then the friction factor coefficient can be determined.

4:01PM G11.00002 Experimental study for turbulent pipe flow at high Reynolds number using LDV - Turbulent intensity profile and influence of measurement volume, NORIYUKI PURIUCHI, National Institute of Advanced Industrial Science and Technology, EISUKE KUSANO, Nagoya Univ., YUKI WADA, Japan Atomic Energy Agency, YOSHIYUKI TSUJI, Nagoya Univ. — We have done new experiments for turbulent pipe flow at high Reynolds number, up to \( Re_{\tauRNA} = 300,000 \) using Hi-Reff. In the experiments, the friction factor was measured by the pressure drop along the pipe and the velocity was measured using LDV. As one of issues of the measurement, the spatial resolution of the measurement is very sensitive for the turbulent intensity for any measurement methods. With increasing Reynolds number, the relative measurement volume of LDV increases. This issue has been discussed for the hot-wire measurement, however still not done for the LDV. In this paper, the influence of the measurement volume of LDV at the wall bounded flow is studied as first. The measurement volume of LDV is controlled by the changing the insertion angle of the laser. Based on the relation between the normalized spatial resolution \( L^* \) and Reynolds number, the criteria for the measurement at high Reynolds number is clarified. Finally, the trend with Reynolds number of turbulent intensity profile for streamwise and wall-normal component are reported. Those data are compared with DNS data.

4:14PM G11.00003 Numerical investigation on very-large-scale motions and the amplitude modulation in the atmospheric boundary layer at very high Reynolds number, HEHE REN, SHIJUN LAIMA, HUI LI, Harbin Institute of Technology, KEY LAB OF SMART PREVENTION AND MITIGATION FOR CIVIL ENGINEERING DISASTERS TEAM — Wall Model Large Eddy Simulations (WMLES) were carried out to investigate the spatial features of very-large-scale motions (VLSMs) in the atmospheric boundary layer flow under different surface roughness at very high Reynolds number, \( Re > 10^6 \). The simulation results display good agreement with field observation and experimental data. By pre-multiplied spectral analysis, the VLSMs reduces or even disappears with increasing roughness, which supports the Bottom-up mechanism indirectly. From the perspective of spatial correlation of flow field, the structural characteristics of VLSMs under various roughness were shown well with observation data. Furthermore, the amplitude modulation (AM) that exerted by the outer layer large-scale motions on the near-wall small-scale motions was discussed. It was found the negative maximum correlation decreases with increasing Reynolds number. And there is an approximate collapse of correlation over different magnitude in Reynolds number when scaled with outer variables. Finally, the physical meaning of the reversal in sign of correlation corresponds to the cross-point of small-scale turbulent intensity and the local peak of energy distribution was found.

1The National Key Research and Development Program of China under grant No. 2018YFC0705605, NSFC under grant No. 51878230

4:27PM G11.00004 Inertial-layer Mean Velocity Profiles for extreme Reynolds number flows, FABIO RAMOS, HAMIDREZA ANBARLOOEI, DANIEL CRUZ, Federal University of Rio de janeiro — For fully developed statistically stationary channel and pipe flows with smooth walls, any averaged quantity of the flow at a distance \( y \) from the wall can be specified by the control parameters \( \nu, \nu_y \), and \( y \). By the Pi theorem, there exist three non-dimensional groups that can be formed from the combination of \( u(y) \) or \( (u(y)) \), and those quantities. In an earlier work, by observing that center of the log-law scales as the geometric mean for \( \delta^{\nu} \), we have proposed an attached eddy framework, which results on a new friction factor formula for extreme Reynolds number, namely \( f \sim \frac{\nu}{Re_{\delta^{\nu}}^{1.75}} \). In this work, by assuming incomplete self-similarity, we show that the new friction factor is compatible with a new MVP formula, namely \( \Delta u(y) = \nu_y(B + (y) \sqrt{y}) \), which in wall units result in the new expression \( u^+ = A(y^{+1/2} + (y) \sqrt{y})^{1/6} \). This formula, with only two free parameters, results in a very good fit for the MVP data obtained from Princeton superpipe experiments, for a large range of extreme Reynolds number, \( Re > 10^6 \), and for a large radial extension above the log-law range.
4:40PM G11.00005 Invariant PDF profile in the log-region of high-Re-number turbulent boundary layer , YOSHIYUKI TSUJI, Nagoya University, ATSUSHI IDO, Railway Technical Research Institute Japan, MICHIKO NISHIOKA, Osaka Prefecture University — The probability density function (pdf) of a streamwise velocity component is studied in zero-pressure gradient boundary layers. From analyzing the data up to $Re_\theta \sim 80000$, it is found that pdfs have self-similar profiles in the log-law region of mean velocity. Pdf profiles asymptote to the universal shape very close to the Gaussian, but are positively skewed at the core region, indicating smaller values in the tail parts. In the log-law region of turbulence intensity, pdf is positively skewed slightly. These characteristics are summarized depending on the Reynolds number.

4:53PM G11.00006 The energy budget at the outer peak of $u'^2$ in turbulent pipe flow.¹, JONATHAN MORRISON, JOSE FERNANDEZ VICENTE, Imperial College — We use the NSTAP data from the Princeton superpipe (Vallikivi PhD thesis 2014) to examine the spatial & spectral energy fluxes close to the outer peak in $u'^2$ in the range of Reynolds numbers, 2.1 $\times$ 10^6 $\leq Re_D \leq 6.0 \times 10^6$, for which the ratio of hot-wire length to Kolmogorov length scale is $l/\eta \approx 10$. Previous results (Hultmark et al. PRL, 108, 2012) suggest that the outer peak emerges at $Re_D \approx 1.1 \times 10^6$, its position exhibiting a locus $y_m^+ = 0.23(Re^{-0.67})$. We note that this is close to the position of the well-known “mesolayer”, which we have also described as the intermediate layer with scaling ($y_m^+ \propto \sqrt{Re} \tau \eta$, where $\eta$ is the rms velocity at $y = \eta$ (Dianw & Morrison, TSFP11 – see also the presentation in the session, “Turbulent Boundary Layers”). It is straightforward to show that the locus of $u_m$ is close to that for the production, $F_m(u'^2)$, where the local-equilibrium approximation approximately holds. Therefore, spectral dynamics are most Kolmogorov-like because spatial transport is minimal. We examine the inertial scaling of the axial velocity spectra and low-order structure functions to explain the importance of intermediate scaling.

5:06PM G11.00007 Thermal transport in hypersonic turbulent boundary layers at high-Reynolds numbers¹, JEAN SANTIAGO, HPCVLab U. of Puerto Rico-Mayaguez, NATHAN TICHENOR, Texas A&M University, GUILLEMROR ARAYA, HPCVLab U. of Puerto Rico-Mayaguez — The evolution of thermal spatially-developing turbulent boundary layers (SDTBL) is studied experimentally and numerically at the hypersonic regime. Experiments were performed in a high-speed blow-down wind tunnel facility located in the National Aerothermochemistry and Hypersonics Laboratory (NAL) at Texas A&M University (TAMU) over a zero-pressure gradient (ZPG) adiabatic flat plate at a Mach number of 4.9 and a Reynolds number of 9,000 based on freestream density, momentum thickness, freestream velocity and wall viscosity. Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) of SDTBL at low and high Reynolds numbers are designed in harmony with experiments. Turbulent inflow information is generated via the dynamic rescaling-recycling approach (J. Fluid Mech., 670, pp. 581-605, 2011), which is extended to compressible flows. DNS and LES results of the field velocity are validated by particle image velocimetry (PIV) experiments. Focus is given to the assessment of Reynolds number on the thermal fluctuations and turbulent heat fluxes, as well as their vertical transport. This presentation will emphasize the calibration effort of the numerical model in order to capture the measured flow structure.

5:19PM G11.00008 Inter-scale modulation in pipe flows at high $Re_c$¹, XIABO ZHENG, YE LI, NAOCE, Shanghai Jiao Tong University, GABRIELE BELLANI, LUCIA MASCOTELLI, ALESSANDRO TALAMELLI, Dipartimento di Ingegneria Industriale, Università di Bologna — Hot-wire measurements of streamwise velocity are conducted in the large-scale pipe-flow facility CicloPe in the friction Reynolds number range $7800 < Re_c < 40 000$. Measurements have been performed both with a rake of 5 synchronized probes arranged at different radial locations, and through radial scans with a single wire traversing the whole pipe radius. Correlation analysis is used to extract geometric features of coherent structures and inter-scale modulation in turbulent pipe flows. The $Re_c$-independence of geometric features is shown. Very-large-scale motions keep inclining and vertical coherence to the wall in the whole pipe radius, while large-scale motions with local coherence gradually become isotropy as reference moves far away from the wall. One-point and two-point amplitude modulation (AM) show that phase difference between large- and small-scales is linear to logarithm of $y/R$ for AM, but is independent on wall-normal location for opposite modulation. In log layer, time-delay for zero-modulation present linear-log relation to $y/R$, but in whole pipe radius, zero-modulation keeps inclining with 4 degree in $\rho^1/\deltam$ map.

¹EPSRC Grant no. EP/I037938

5:19PM G11.00008 Inter-scale modulation in pipe flows at high $Re_c$¹, XIABO ZHENG, YE LI, NAOCE, Shanghai Jiao Tong University, GABRIELE BELLANI, LUCIA MASCOTELLI, ALESSANDRO TALAMELLI, Dipartimento di Ingegneria Industriale, Università di Bologna — Hot-wire measurements of streamwise velocity are conducted in the large-scale pipe-flow facility CicloPe in the friction Reynolds number range $7800 < Re_c < 40 000$. Measurements have been performed both with a rake of 5 synchronized probes arranged at different radial locations, and through radial scans with a single wire traversing the whole pipe radius. Correlation analysis is used to extract geometric features of coherent structures and inter-scale modulation in turbulent pipe flows. The $Re_c$-independence of geometric features is shown. Very-large-scale motions keep inclining and vertical coherence to the wall in the whole pipe radius, while large-scale motions with local coherence gradually become isotropy as reference moves far away from the wall. One-point and two-point amplitude modulation (AM) show that phase difference between large- and small-scales is linear to logarithm of $y/R$ for AM, but is independent on wall-normal location for opposite modulation. In log layer, time-delay for zero-modulation present linear-log relation to $y/R$, but in whole pipe radius, zero-modulation keeps inclining with 4 degree in $\rho^1/\deltam$ map.

¹This work was supported by EuHIT (No. 312778), and Natural Science Foundation of China (Nos. 51761135012/51479114/11872248).


3:48PM G12.00001 Honoring Ted OBrien: High order methods for filtered and probability density function models , GUSTAAF JACOBS, San Diego State University — The systems of partial differential equations that govern probability and filtered density function models can be solved directly using numerical methods. Oftentimes, this type of system is also solved using a combination of Monte-Carlo and stochastic differential equations. If the density function model is coupled with another model that has feedback, as can occur in multi-physics or multi-phase environments, then the numerical coupling must be consistent for both approaches to obtain an accurate numerical solution. In this talk, I will discuss recent progress in the development of high order accuracy methods for models governing, chemically reaction and/or particle-laden, high-speed flows with shocks. High order distribution functions and weighted interpolations combined with spectral elements methods are presented and are shown to give accurate results for time-dependent problems that require long time integration.
the turbulent strain-rate and the destruction of iso-surface area due to the combined effects of diffusion and surface curvature. The T/NT interface in a turbulent free shear flow, and a passive scalar field which represents an inert tracer such as dye concentration or the vorticity magnitude, which represents in its transport equation. Iso-surface properties, such as the surface area, are evaluated by converting the surface integrals to volume integrals.

FOLUSO LADEINDE, Stony Brook University — The authors initial studies on compressible turbulence and combustion reported.

variants so that stochastic methods are required for its solution; in the present work the stochastic fields method is utilised. It has been applied to simulate the evolution of self-excited thermo-acoustic instabilities in a gas turbine model combustor, using a fully compressible formulation. An unstable operating condition in the PRECCINSTA combustor, involving flame oscillation driven by thermo-acoustic instabilities, is the chosen target configuration. The flame’s self-excited oscillatory behaviour is successfully captured without any external forcing being involved. Power spectral density analysis of the oscillation reveals a dominant thermo-acoustic mode at a frequency of 30 Hz providing remarkably good agreement with experimental observations. Moreover, the predicted limit-cycle amplitude closely matches the experimental value obtained with rigid metal combustor side walls.

2nd author: D. Fredrich, Imperial College London

4:14PM G12.00003 Physics-Based vs. Data-Driven Modeling for Turbulence and Combustion, SHARATH GIRIMAJI1,2, Ocean and Aerospace Engineering, Texas AM University — Honoring Ted O’Brien: Ted O’Brien had a long and distinguished career in modeling and computing chemically reacting turbulent flows. He made important contributions toward modeling/computation of auto-ignition in turbulent mixtures, conditional scalar dissipation, PDF (probability density function) methods and mapping closure methods. Currently, drive toward use of data-driven models is pervasive in nearly all fields involving complex phenomena including turbulent combustion. This presentation will discuss some of the benefits and challenges of using data-driven models for prediction of reacting turbulent flows. For a variety of turbulence and combustion features, we will compare the strengths and weaknesses of data-driven modeling against that of physical modeling. Specifically will we examine the general capabilities of data-driven approaches for handling (i) distant interactions – specifically non-local effects due to the elliptic nature of pressure and (ii) purely local process of chemical reactions. The talk will conclude with some recommendations on synergistically combining physics-based and data-driven approaches for developing predictive tools for turbulence and combustion.

1This paper is being submitted to: Session 31.5 (Honoring Ted O’Brien)

4:27PM G12.00004 Differential diffusion modelling in transported PDF simulations of turbulent flames1, ZHUYIN REN, Tsinghua University, HUA ZHOU, University of New South Wales, TIANWEI YANG, Tsinghua University — Honoring Ted O’Brien. The modelling strategy to incorporate differential diffusion effects in transported density function method (PDF), particularly in the context of large eddy simulation (LES) is proposed. Differential diffusion at the filter scale is resolved by the mean drift term in composition equations, while subgrid differential diffusion is modelled by the augmented mixing models that can account for differential mixing rates for each individual species. Both RANS/PDF and LES/FDF simulations of a jet-in-hot-coflow methane-hydrogen flame have been performed to investigate the effects of differential diffusion on flame characteristics.

1This work is supported by National Natural Science Foundation of China 91841302 and 91441202.

4:40PM G12.00005 Mathematical Models For Eulerian Conditional Statistics in a Complex Turbulent Flow, JAMES HILL, Iowa State University (Retired), EMMANUEL HITIMANA, MICHAEL OLSEN, RODNEY FOX, Iowa State University — Honoring Ted O’Brien. Conditional moment closure (CMC) methods were developed for predicting turbulent reacting flows. However, conditional averages appear as unclosed terms that need to be modeled. In the present work the linear approximation and PDF gradient models were used to predict the conditional mean velocity and mixture fraction and compare with experimental data obtained for a macroscale multi-inlet vortex chemical reactor (macro-MIVR) using laser diagnostic techniques. The results for velocity conditioned on mixture fraction show that the linear model works well in a low turbulence region away from the reactor center. The PDF model with an isotropic turbulent diffusivity performs poorly for the tangential and axial conditional velocities, but a modified version that considers three components of turbulent diffusivity is better. Furthermore, the mixture fraction conditioned on the velocity vector components has a more linear behavior near the reactor center, where the probability density function (PDF) of the mixture fraction is close to a Gaussian distribution.

4:53PM G12.00006 On the kinematics of scalar iso-surfaces in a turbulent, temporally developing jet, BRANDON BLAKELEY, WEIRONG WANG, DUANE STORTI, JAMES RILEY, University of Washington — The kinematics and dynamics of scalar iso-surfaces in turbulent flows is of fundamental importance for a number of problems, e.g., the stoichiometric flame surface in non-premixed combustion or the turbulent/non-turbulent interface in turbulent shear flows. We investigate the effects of turbulence on iso-surfaces by examining the surface area density, Σ, and its evolution. Using direct numerical simulation of a temporally developing jet and a novel algorithm for evaluating iso-surface properties, we report on the direct computation of Σ and the terms in its transport equation. Iso-surface properties, such as the surface area, are evaluated by converting the surface integrals to volume integrals on a regularly-sampled grid. In particular, we analyze the behavior of two different scalar iso-surfaces: the vorticity magnitude, which represents the T/NT interface in a turbulent free shear flow, and a passive scalar field which represents an inert tracer such as dye concentration or the mixture fraction. Differences between the evolution of the two iso-surfaces will be addressed, such as the production of iso-surface area due to the turbulent strain-rate and the destruction of iso-surface area due to the combined effects of diffusion and surface curvature.

5:06PM G12.00007 Investigation of Two-Phase Supersonic Combustion in Hypersonic Flight, FOLUSO LADEINDE, Stony Brook University — The authors initial studies on compressible turbulence and combustion in high-speed flows were done via DNS in collaboration with the Late Professor Edward E. O’Brien in several joint publications on the topic. However, the authors focus has evolved, and the transport of momentum, energy, and chemical species in supersonic spray combustion for systems that approximate the scramjet engine in hypersonic flight is of current interest. Many advantages can be derived from the use of liquid fuels, such as the higher heat release and ease of storage and handling. The system in question is complicated by the interaction of many effects, including those due to combustion, evaporation, turbulence, shock waves, and their interactions. Consequently, many studies have addressed these issues. Based on the parameters for the application of interest, the point-particle approach via the Eulerian-Lagrangian formulation is followed in the present endeavor. This approach introduces explicit force and energy sources, some of which involve history integrals. The significance of these sources is investigated in terms of their roles in the rather complex drop breakup mechanism in the presence of shockwaves, and the eventual evaporation and combustion to provide the needed propulsive force. The progress made by the author will be reported.
The resulting modifications of vertical heat transport and solute concentration are investigated at varying shear rates. fingering instability, leading to highly anisotropic DDC dynamics associated with the formation of regular sheets of the slowly-diffusing scalar. U. of North Carolina pipes 1 groups, such as the Rossby number, the Froude number, and the Bingham number. a critical transition, leads to a complete removal of the displaced fluid. The results are quantitatively analyzed using the flow dimensionless experimental parameters, such as the pipe inclination angle, the density difference, and most importantly the pipe rotation speed. For both speed imaging techniques. The analysis of our experimental results reveals that the displacement process can be affected by variations of the yield stress fluid. The experiments are performed in a long transparent pipe so that the displacement flow patterns can be visualized using high speed imaging techniques. The experimental results can be affected by variations of the experimental parameters, such as the pipe inclination angle, the density difference, and most importantly the pipe rotation speed. For both Newtonian and yield stress displacements, increasing the pipe rotation speed enhances the transverse mixing between the fluids and, above a critical transition, leads to a complete removal of the displaced fluid. The results are quantitatively analyzed using the flow dimensionless groups, such as the Rossby number, the Froude number, and the Bingham number.

1This research has been supported financially by the Fonds de Recherche du Quebéc Nature et technologies and the Canada Foundation for Innovation. S.L. acknowledges the Ph.D. scholarship provided by the China Scholarship Council.

3:48PM G13.00001 Buoyant displacement flows of miscible fluids in axially rotating pipes 1. SEYED MOHAMMAD TAGHAVI, SHAN LYU, Universit Laval — This work experimentally considers the effects of a pipe axial rotating motion on buoyant displacement flows of miscible fluids in an inclined pipe. A heavy Newtonian fluid displaces a light Newtonian or yield stress fluid. The experiments are performed in a long transparent pipe so that the displacement flow patterns can be visualized using high speed imaging techniques. The experimental results can be affected by variations of the experimental parameters, such as the pipe inclination angle, the density difference, and most importantly the pipe rotation speed. For both Newtonian and yield stress displacements, increasing the pipe rotation speed enhances the transverse mixing between the fluids and, above a critical transition, leads to a complete removal of the displaced fluid. The results are quantitatively analyzed using the flow dimensionless groups, such as the Rossby number, the Froude number, and the Bingham number.

4:01PM G13.00002 Nonnormal transient growth of triadic resonant internal gravity waves, KEVIN HA, JEAN MARC CHOMAZ, Ecole Polytechnique, SABINE ORTIZ, ENSTA — Triadic resonant instability is a key component in the understanding of the dissipation process of inertia gravity waves for geophysical applications like oceanic circulation, which is still incompletely understood. A key process in the oceanic circulation is the vertical mixing, which makes it possible for dense deep water to reach the surface. Global warming regulation by the ocean then depends on mechanisms controlling the vertical mixing of deepwater masses. It was recently predicted by Garrett & Kunze (2007) that triadic resonances are responsible for inertial gravity waves generated by interaction between the barotropic tide and global topography (continental shelf, underwater mount). The present work focuses on the energy approach of the triadic resonant instability and demonstrates that due to the nonnormality of the evolution operator, stable triadic resonant interactions result in a transient amplification of perturbation energy. Computations show that they can lead to a longer and more intense transient growth than unstable triads. Instead of being related to the differential growth of a stable and an unstable modes like for unstable triads, the transient growth of stable triads originates from the differential rotation (i.e. phase shift) of two stable eigenmodes.

4:14PM G13.00003 Mechanisms of drag enhancement on bodies settling in a linearly stratified fluid, JACQUES MAGNAUDET, Institut de Mecanique des Fluides de Toulouse, CNRS and Universite de Toulouse, France, JIE ZHANG, State Key Laboratory for Strength and Vibration of Mechanical Structures, School of Aerospace, Xian Jiaotong University, China, MATHIEU MERCIER, Institut de Mecanique des Fluides de Toulouse, CNRS and Universite de Toulouse, France — Using DNS, we compute the flow past a sphere settling with constant speed in a linearly stratified fluid over a wide range of Reynolds, Prandtl and Froude numbers. Thanks to a rigorous mathematical decomposition procedure, we identify the various contributions to the stratification-induced drag, especially that due to the dragging of light fluid around the body and within its wake, and that induced by the distortion of the vortex lines. Combining DNS results and scaling laws provided by the dominant balances in the density and momentum equations, we show how these two contributions vary with the control parameters. It turns out that for large Prandtl numbers and Reynolds numbers up to $O(10^4)$, the drag enhancement is essentially due to the specific structure of the vorticity field set in by buoyancy effects, while effect of the dragging of light fluid near the body surface takes over for larger Reynolds numbers.

4:27PM G13.00004 Decay of convective boundary layer and effect of decreasing rate of surface heat flux, MINGXIANG ZHAO, LIAN SHEN, University of Minnesota — Decay of convective boundary layer (CBL) is a model of the late-afternoon transition of the unstably stratified atmospheric boundary layer. In this study, simulations were performed to investigate the decay process under different decreasing rates of the surface heat flux. We decomposed the turbulence fluctuations into different scales to investigate their responses to the decreasing surface heat flux. It is shown that the decreasing rate of the large-scale vertical velocity fluctuations is much greater than that of the small-scale fluctuations regardless how fast the decreasing rate of the surface heat flux is. Moreover, the large-scale fluctuations experience several stages of temporary enhancements due to the local upward heat flux, while the small-scale fluctuations decreases monotonously. For the decay process of temperature fluctuations, the present study showed essential differences from that of small-scale fluctuations. For instance, the temperature fluctuations respond to the decreasing surface heat flux more quickly and the small-scale fluctuations decay faster than the large-scale fluctuations.

4:40PM G13.00005 Influence of shear on fingering dynamics in double diffusive convection, PEJMAN HADI SICHANI, CRISTIAN MARCHIOLI, University of Udine, FRANCESCO ZONTA, ALFREDO SOLDATI, TU Wien — We examine the effect of shear on double-diffusive convection (DDC) in a confined fluid layer under the Oberbeck-Boussinesq condition. DDC is characterized by the competing action of a stably-stratified, rapidly-diffusing scalar temperature and an unstably-stratified, slowly-diffusing scalar solute concentration. This problem has five governing parameters: The thermal Prandtl number, $Pr_T$ (momentum to thermal diffusivity ratio); the thermal Rayleigh number, $Ra_T$ (measure of the fluid instability due to temperature and density differences); the Lewis number, $Le$ (thermal to solute diffusivity ratio); the density stability ratio, $R_i$ (measure of the effective flow stratification), and the shear rate, $\Gamma$. We investigate double-diffusive fingering subject to uniform shear in a wall-bounded domain by performing a campaign of highly-resolved numerical simulations in the $(Pr_T, Ra_T, Le, R_i, \Gamma)$ parameter space. Preliminary results show that shear tends to dampen the growth of fingering instability, leading to highly anisotropic DDC dynamics associated with the formation of regular sheets of the slowly-diffusing scalar. The resulting modifications of vertical heat transport and solute concentration are investigated at varying shear rates.
4:53PM G13.00006 Estimation of mixing in a lock-exchange flow using molecular tagging velocimetry and thermometry, TANMAY AGRAWAL, JIMMY PHILIP, JOE KLEWICKI, The University of Melbourne — Gravity currents produced by a lock-exchange experiment are studied using the single-component version of molecular tagging velocimetry (MTV) in conjunction with its thermal counterpart, molecular tagging thermometry (MTT). The experiments are conducted in a Perspex tank of 2.0 m x 0.2 m x 0.2 m where the lock is located mid-way. Therefore, the current is studied only during the slumping phase and there are no transitions associated with the end-wall reflection. For these experiments, the initial density difference is created by introducing a thermal inhomogeneity on either side of the lock as compared to the general experimental practice of dissolving a salt on one side. The flow is first visualized by mixing a dye on the heavier side to fully distinguish the experimental parameters. Subsequently, MTV, MTT images are acquired that contain approximately 1000 data points distributed across the interface of hot and cold fluid. This high-resolution velocity and temperature data is then used to quantify the mixing being taken place at the interface. Specifically, background potential energy of the flow is evaluated over time to estimate the extent of irreversible mixing while an equivalent Thorpe scale is calculated to estimate the size of overturning eddies.

5:06PM G13.00007 Wall-bounded stably-stratified turbulence at high Reynolds number, FRANCESCO ZONTA, Institute of Fluid Mechanics and Heat Transfer, TU Wien, PEJMAN HADI SICHANI, Polytechnic Department, University of Udine, ALFREDO SOLDATI, Institute of Fluid Mechanics and Heat Transfer, TU Wien — Wall-bounded stably stratified turbulence is encountered in many industrial and natural processes. Examples include fluid motion in heat transfer devices or transport/mixing of organic species in terrestrial water bodies. In this work, we focus on stably stratified turbulent channel flow at high shear Reynolds number Reτ. We perform a campaign of pseudo-spectral direct numerical simulations (DNS) of the governing equations, written under the Boussinesq approximation, in the shear Richardson number space Riτ = Gr/Reτ^2 (which measures the relative importance of buoyancy compared to inertia, with Gr the Grashof number). In particular, we fix the Reynolds number Reτ = 1000 and we change Gr so to cover a broad range of Riτ values. For increasing Riτ, turbulence is sustained only near the boundaries, whereas non-turbulent wavy structures (Internal Gravity Waves, IGW), also favored by the presence of intermittent bursts, are observed at the core of the channel. Naturally, the presence of IGW alters the overall transfer rates of momentum and heat, as well as the mixing efficiency of the flow. We believe that the present results may give important contributions to future turbulence parametrization and modeling in this field.

5:19PM G13.00008 Intermittency and modal dynamics of stratified mixing at oceanographic scales, ALBERTO SCOTTI, University of North Carolina at Chapel Hill, PIERRE-YVES PASSAGGIA, Prisme Lab. University of Orleans — Stratified turbulent channel flows are known to generate strongly intermittent dynamics where the flow alternates irregularly between the growth of stratified shear instabilities, regions of decaying turbulence, and quasi laminar states. Quantifying intermittency at oceanic scales from field measurements or numerical simulations may become intractable because of sampling issues. The former may offer too few spatial measurements while the latter may prove too computationally expensive. Laboratory experiments provide an interesting alternative and allow for circumventing both difficulties. This talk therefore explores this issue with the aid of an experimental dataset acquired in the UNC Joint Fluid Lab. In a large tank, we generate shear-driven turbulent mixing that spans a significant range of the parameter space encountered in the ocean and for several thousands of eddy turnover times. Hot wire and conductivity measurements are paired with PIV and LIF measurements in a plane to extract the modal features of the flow and their intermittency. These measurements are finally compared with measurements of the bulk Richardson number which is found to be key driver of these dynamics. We finally discuss the implication for the interpretation of field data.

1The authors acknowledge the support of the National Science Foundation Grant Number OCE-1736989.


3:48PM G14.00001 Do steady rolls maximize heat transport in truncated models of Rayleigh-Bénard convection?1, DAVID GOLUSKIN, University of Victoria, CHARLES R. DOERING, University of Michigan, ANUJ KUMAR, University of California, Santa Cruz, MATTHEW OLSON, University of Michigan — In Rayleigh-Bénard convection, steady rolls are the simplest nonlinear states. They exist for all Rayleigh numbers (Ra) above the primary instability but are unstable at large Ra. Heat transport by steady rolls is comparable to that by turbulent convection if the aspect ratio of rolls is varied with Ra to maximize heat transport (Waleffe et al., Phys. Fluids 27, 2015). Here we ask: do steady rolls transport more heat than any other flows, including turbulent or unstable time-periodic flows? Answering this question in the affirmative requires computing exact upper bounds on heat transport, and showing that steady rolls saturate the bounds. Exact bounds for the governing PDEs are beyond current capabilities, so we instead study ODE models of increasing dimension, each derived as a Galerkin truncation of the PDEs governing 2D convection between free-slip plates. For the ODEs we compute sharp upper bounds on heat transport by using polynomial optimization. In particular, upper bounds are provided by solutions to convex optimization problems, whose constraints require certain expressions to be sums of squares of polynomials, and which are solved numerically using semidefinite programming. To find the states that saturate these bounds, we perform numerical bifurcation analysis.

1Supported by NSERC awards RGPIN-2018-04263 and RGPAS-2018-522657, and NSF award DMS-1813003

4:01PM G14.00002 Numerical study of thermal transfer in Rayleigh-Bénard convection under rarefied gas conditions, GIANLUCA DI STASO, BIJAN GOSHAYESHI, FEDERICO TOSCHI, HERMAN CLERCX, Eindhoven University of Technology — The Rayleigh-Bénard problem has been analyzed in depth theoretically, experimentally and numerically under a broad range of conditions such as small and large Prandtl number, high Rayleigh number turbulent convection, non-altregnate compressibility effects, as well as under rotation. The large majority of these studies have in common the underlying assumption of a continuum flow, i.e. the molecular mean free path is much smaller than any characteristic macroscopic spatial scale of the flow. In this contribution, we depart from this assumption and we numerically study the final state of a 2D Rayleigh-Bénard system under rarefied gas conditions using the Direct Simulation Monte Carlo method (DSMC), the standard particle-based numerical method for simulating rarefied gas flows. By collecting a large number of statistical samples, we quantitatively measure the heat flux enhancement when convection is present and we determine the influence of rarefaction conditions on the maximum attainable heat flux. Finally, we show that onset of convection is found only for a limited range of Rayleigh number and this range is reduced as the degree of rarefaction increases.
4:14PM G14.00003 Transition of the flow dynamics in two-dimensional Rayleigh-Bénard convection1. ZHENYUAN GAO, Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, YUN BAO, Department of Mechanics, Sun Yet-sen University, SHIDU HUANG, Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology — We investigate the flow dynamics in two-dimensional Rayleigh-Bénard convection through high resolution direct numerical simulation, with the Rayleigh number Ra range being $10^7$ to $10^9$ and the Prandtl number Pr fixed at 4.3. It is found that there exists a transitional Rayleigh number $Ra_c$ at which the flow pattern changes significantly and the large-scale circulation (LSC) evolves from an elliptical shape into a circular one. Detailed Fourier mode analysis reveals that, while the single-roll mode becomes weaker and other modes become stronger during the transition, all the flow modes experience violent fluctuations. This is also manifested by the sharp change of the local turbulent fluctuations near $Ra_c$, in both magnitude and Ra-dependent scaling. We understand this transition by stability analysis.

1This work was supported by the National Natural Science Foundation of China under Grant Nos. 11702128, 91752201 and 11772362.

4:27PM G14.00004 Using Persistent Homology to Compare Chaotic Dynamics Between Experiments on and Simulations of Rayleigh-Bénard Convection1. BRETT TREGONING, Georgia Institute of Technology, SAIKAT Mukherjee, Virginia Polytechnic Institute and State University, RACHEL LEVANGER, University of Pennsylvania, MU XU, Columbia University, JACEK CYRANKA, University of California San Diego, KONSTANTIN MISCHAIKOW, Rutgers University, MARK PAUL, Virginia Polytechnic Institute and State University, MICHAEL SCHATZ, Georgia Institute of Technology — Persistent homology is a tool from algebraic topology that can be used to efficiently detect pattern features in image data. In the spatio-temporally chaotic flow known as spiral defect chaos in Rayleigh-Bénard convection, we explore the use of pattern features detected by persistent homology as a proxy for fundamental dynamical quantities that are not observable in experimental data but can be calculated from simulations, such as leading-order Lyapunov vectors. In simulations, we have identified that convective plumes are highly correlated with the leading-order Lyapunov vectors; however, we find that plumes appear in experiments at distinctly different rates than for Boussinesq simulations at the same parameter values. We describe work to resolve this discrepancy by accounting for non-Boussinesq effects in both experiments and simulations.

1This work is supported under NSF grant DMS 1622113, DMS 1125302, CMMI-1234436, and DARPA HR0011-16-2-0033.

4:40PM G14.00005 Marginally stable Rayleigh–Bénard convection. BAOLE WEN, University of Michigan, ZIJING DING, University of Cambridge, GREGORY CHINI, University of New Hampshire, RICH KERSWELL, University of Cambridge, UNIVERSITY OF MICHIGAN COLLABORATION, UNIVERSITY OF NEW HAMPSHIRE COLLABORATION, UNIVERSITY OF CAMBRIDGE COLLABORATION — We propose a new strategy to predict the heat transport in 2D Rayleigh–Bénard convection between stress-free isothermal boundaries. The Constantin–Doering–Hopf (CDH) variational framework, in which the temperature is decomposed into a background profile plus a fluctuation field and the background profile is required to satisfy a marginal energy-stability constraint, provides a formalism for determining boundaries. In the spatio-temporally chaotic flow known as spiral defect chaos in Rayleigh–Bénard convection, we explore the use of pattern features detected by persistent homology as a proxy for fundamental dynamical quantities that are not observable in experimental data but can be calculated from simulations, such as leading-order Lyapunov vectors. In simulations, we have identified that convective plumes are highly correlated with the leading-order Lyapunov vectors; however, we find that plumes appear in experiments at distinctly different rates than for Boussinesq simulations at the same parameter values. We describe work to resolve this discrepancy by accounting for non-Boussinesq effects in both experiments and simulations.

4:53PM G14.00006 Effect of aspect ratio on the Rayleigh convection of Maxwell viscoelastic fluids in a cavity heated from below1. ILDEBRANDO PEREZ-REYES, Universidad Autonoma de Chihuahua, ALEJANDRO SEBASTIAN ORTIZ-PEREZ, Universidad Autonoma de Baja California, NESTOR GUTIERREZ-MENDEZ, Universidad Autonoma de Chihuahua, FLUIDS, HEAT AND MASS TEAM — Interest results, of the effect the parity of convective rolls, on the Rayleigh convection in a Maxwell viscoelastic fluid confined in a 2D cavity are presented. A linear stability analysis have shown how the fluid stability changes for different values of the aspect ratio ranging from 0.1 to 10. It was found that vertical and horizontal parity of the temperature and velocity solutions, related to parity of the number of rolls distributed horizontally or vertically, gives different stability scenarios. On the other hand, kinks also appear in the curves of criticality, which is also found in the stability of newtonian fluids which also depend on the symmetry of the solutions. Results and physical mechanisms shall be presented in terms of plots of the critical Rayleigh number and the frequency of oscillation for different cases encompassing perfect thermal conducting, or insulating, lateral, or horizontal, walls. Different values of the dimensionless relaxation time $F$ are considered to discuss the physical mechanism of instability for fixed values of the Prandtl number $Pr$ as well.

1Funding due to CONACyT through CB-2015-01 project 255839 is acknowledged.

5:06PM G14.00007 Thermal convection over a fractal surface1. SRIRKANT TOPPALADOODDI, ANDREW WELLS, University of Oxford, CHARLES DOERING, University of Michigan, JOHN WETTLAUFER, Yale University & Nordita — We use well resolved numerical simulations to study Rayleigh–Bénard convection in cells with a fractal boundary in two dimensions for $Pr = 1$ and $Ra \in [10^7, 2.15 \times 10^9]$. The fractal boundaries are functions characterized by power spectral densities $S(k)$ that decay with wavenumber as $S(k) \sim k^{-p}$ ($p < 0$). The degree of roughness is quantified by the exponent $p$ with $p < -3$ for smooth (differentiable) surfaces and $-3 \leq p < -1$ for rough surfaces with Hausdorff dimension $D_H = \frac{1}{2} (p + 5)$. By computing the exponent $\beta$ in power law fits $Nu \sim Ra^\beta$, where $Nu$ and $Ra$ are the Nusselt and the Rayleigh numbers, we observe that heat transport increases with roughness. For $p = -3.0$, $-2.0$ and $-1.5$ we find, respectively, $\beta = 0.256$, $0.281$ and $0.306$. For a given value of $p$ we observe that the mean heat flux is insensitive to the details of the roughness.

1Work supported by a Research Fellowship from All Souls College Oxford, US National Science Foundation award DMS-1813003, and Vetenskapsrådet No. 638-2013-9243.
India.

coordinate leading to universal overlap description in terms of the intermediate variable. Spectra and correlation maps also show promising
the mean velocity profile shows that the Reynolds-number-dependence of the overlap layer can be effectively absorbed into an intermediate
hotwire and PIV data from our wall jet setup, over a range of Reynolds numbers, to substantiate this view. Further, an overlap analysis of
that the wall-jet flow structure consists of universal full free-jet outer flow and wall-scaled inner flow. We further argue that there exists strong,
wall jets find important applications in engineering and meteorology, and present unique features such the non-monotone mean velocity profile,

1

1 analysis is based on highly resolved direct numerical simulations.

functions on an external intermittency parameter, it is shown that the self-similarity solution of structure functions can be recovered. The
provided that this departure is primarily due to external intermittency and the associated break-down of self-similarity. By conditioning structure
of structure functions exhibits a growing departure from the prediction of classical scaling solutions toward the edge. Empirical evidence is
on small-scale turbulence is studied across the jet by the self-similarity solution of higher-order structure functions. It is shown that the scaling
turbulent interface, that separates the inner turbulent core from the outer irrotational surrounding fluid. The impact of external intermittency
turbulent jet flow. In turbulent jet flows, the phenomenon of external intermittency originates from a very thin layer, known as turbulent/non-
effect of internal and external intermittency on the statistical properties of small-scale turbulence is investigated in a temporally evolving, planar

3

We span Reynolds numbers from 7 to 25 x10^3

and nominally 2D flow structures, while varying jet size, speed, height and angle, and fluid properties such as surface tension and viscosity.

3:48PM G15.00001 How is the eigenframe of the rate-of-strain tensor perturbed by density gradients? . DOMINIQUE FRATANTONIO, CHRIS LAI, JOHN CHARONKO, KATHY PRESTRIDGE, Los Alamos National Laboratory — We present an experimental study on a variable-density turbulent jet. The jet’s three-dimensional density and velocity fields have been reconstructed using a time-resolved simultaneous stereoscopic PIV and planar LIF system together with the Taylor’s frozen turbulence hypothesis. The obtained dataset is then analyzed to understand how density gradients perturb the eigenframe of the rate-of-strain tensor whose alignments with the vorticity vector govern the key turbulence process of vortex-stretching.

4:01PM G15.00002 Air entrainment by 2D plunging jets1 . NICOLE FARIAS, SIMO A. MKIHARJU , UC Berkeley — Air entrainment from liquid plunging jets are commonly found in nature and industry applications. As a result, this phenomenon has been the focus of many studies in the past, which showed it to have a significant contribution to the energy and mass transfer between gas and liquids. Air entrainment was also discovered to be a function of various parameters, such as jet impact velocity, geometry, fluid properties, and resulting jet instabilities. Due to the complex multiphase flow involved, no exhaustive characterization or theory is yet available, and limited data has been published on the dynamics within the entrainment zone once entrainment becomes significant enough to make the flow optically opaque. We study 2D plunging jets using high-speed video, point probes and O(10kHz) 2D X-ray densitometry to examine the air entrainment and nominally 2D flow structures, while varying jet size, speed, height and angle, and fluid properties such as surface tension and viscosity. We span Reynolds numbers from 7 to 25 x10^3, and Weber numbers from 0.3 to 40.4. The high-speed X-ray densitometry we employ enables time-resolved analysis of phase fraction in large areas with greater efficiency and detail than feasible utilizing point probes alone.

1This work has been supported by the American Bureau of Shipping and by the Society of Naval Architects and Marine Engineers.

4:14PM G15.00003 The combined effect of internal and external intermittency in turbulent jet flows1 . MICHAEL GAUDING, YACINE BRAHAMI, LUMINITA DANAILA, EMILEN VAREA, CORIA — The combined effect of internal and external intermittency on the statistical properties of small-scale turbulence is investigated in a temporally evolving, planar turbulent jet flow. In turbulent jet flows, the phenomenon of external intermittency originates from a very thin layer, known as turbulent/non-turbulent interface, that separates the inner turbulent core from the outer irrotational surrounding fluid. The impact of external intermittency on small-scale turbulence is studied across the jet by the self-similarity solution of higher-order structure functions. It is shown that the scaling of structure functions exhibits a growing departure from the prediction of classical scaling solutions toward the edge. Empirical evidence is provided that this departure is primarily due to external intermittency and the associated break-down of self-similarity. By conditioning structure functions on an external intermittency parameter, it is shown that the self-similarity solution of structure functions can be recovered. The analysis is based on highly resolved direct numerical simulations.

1Financial support was provided by EMCO2RE and FIVATHE.

4:27PM G15.00004 Structure of two-dimensional turbulent wall jets1 . SHIVSAI DIXIT, HARISH CHOUDHARY, ABHISHEK GUPTA, THARA PRABHAKARAN, Indian Institute of Tropical Meteorology (IITM), Pune, Maharashtra, India, ABHAY KUMAR SINGH, Institute of Science, Banaras Hindu University, Varanasi, Uttar Pradesh, India — Two-dimensional turbulent wall jets find important applications in engineering and meteorology, and present unique features such as the non-monotone mean velocity profile, a region of counter-gradient momentum diffusion etc. that are very different from other canonical turbulent wall-bounded flows. We propose that the wall-jet flow structure consists of universal full free-jet outer flow and wall-scaled inner flow. We further argue that there exists strong, nonlinear inner-outer interaction which could lead to the Reynolds number dependence of the overlap layer in wall jets. We present experimental hotwire and PIV data from our wall jet setup, over a range of Reynolds numbers, to substantiate this view. Further, an overlap analysis of the mean velocity profile shows that the Reynolds-number-dependence of the overlap layer can be effectively absorbed into an intermediate coordinate leading to universal overlap description in terms of the intermediate variable. Spectra and correlation maps also show promising support for the proposed structure of wall jets.

1Authors gratefully acknowledge the support from the Director, IITM, Pune and Ministry of Earth Sciences (MoES), Government of India.
4:40PM G15.00005 Optical measurement of the interaction between outwardly oriented, steady gas jets. FRANK AUSTIN MIER, SIMONE HILL, MICHAEL HARGATHER, New Mexico Tech — Nearby gas jets issuing into a quiescent environment interact with each other in various manners dependent on their relative spacing and orientation. While the study of a single turbulent jet is classical to fluid dynamics, and entrainment between parallel jets at varying spacing has been quantified, this investigation aims to determine how an outward projection will affect the jet interaction. Here, Particle Image Velocimetry (PIV) and high-speed schlieren imaging are performed to characterize this interaction. An experiment was designed to produce steady gas venting through custom orifice plates at a range of choked and un-choked stagnation pressures. Orifice plates produced jets offset at incremented angles from 0° to 60°. Downstream jet interactions and flow features were measured through high-speed schlieren images. The velocity profile findings are compared to the well-defined parallel jet case. This work provides a fundamental understanding of the flow structure and velocity field of outwardly directed jets with broad potential application, including the external fluid-dynamics of lithium ion battery venting failures.


4:53PM G15.00006 Bifurcating jet in transverse acoustic field. EIRIK ASOY, PhD, JOSE G. AGUILAR, Post-doctoral, NICHOLAS A. WORTH, JAMES R. DAWSON, Professor — The far-field of an axisymmetric round jet is usually characterized by the momentum flux at the nozzle exit. However, it has been shown that when the jet is submitted to external forcing, the far-field exhibits different behavior from a classical jet. We present experiments of an axisymmetric jet subjected to external forcing by placing the nozzle exit in a standing acoustic field by forcing a rectangular box with loud-speakers. Several forcing conditions at the nozzle exit, i.e., differences of transverse and longitudinal velocity oscillations, are actuated by placing the nozzle at different positions relative to the pressure node. Time-resolved Particle Image Velocimetry (PIV) combined with microphone measurements are used to characterize the flow field in a plane of interest and the acoustic fields. It is found that the jet bifurcates, i.e., splits into two or three separate momentum streams, at sufficiently high forcing levels when placed anywhere else than at a pressure anti-node. Furthermore, the flow field is asymmetric at any position between the pressure and velocity nodes. This asymmetry is symmetric across the pressure anti-node which is shown to be linked to the relative phase between transverse and longitudinal velocity oscillations.

5:06PM G15.00007 Reevaluating the jet breakup regime diagram. BEN TRETTEL, University of Texas at Austin — Identifying the regime of a liquid jet is necessary to determine the physical mechanisms causing breakup and consequently how to model the jet. Existing regime diagrams are based on a small amount of data classified by superficial visual characteristics, making these diagrams too inaccurate to reliably determine the correct regime. A more accurate regime diagram is developed using a large compilation of breakup length data combined with theory where the data is sparse. Improvements in the regime diagram include a new regime, the addition of the nozzle critical Reynolds number and the turbulence intensity as variables, and the recognition that how the regimes change with increasing velocity (i.e., Rayleigh to first wind-induced to second wind-induced to atomization) is not universal.

5:19PM G15.00008 Droplet size distribution along the near-field interface of immiscible turbulent jets. ERIC IBARRA, FRANKLIN SHAFFER, OMER SAVAS, UC Berkeley — This work examines the droplet distributions of submerged, immiscible, turbulent jets in the near field. Experiments consist of silicone oil jets of two viscosities submerged in a water tank. The jet Reynolds numbers are in the range of Re ∼ 3,500 – 27,000. Shadowgraphy is used to investigate the droplet sizes at the edge of the jets which are quantified using Hough transformation. The droplet size distribution results are observed to be bi-modal. The form of these distributions draw attention to prevalence of observed small satellite droplets being formed from the rupture of shear-layers in the flow. Results suggest that the radii of the small satellite droplets are independent of Re, while the average size of the large droplets is governed by the Weber number. High speed videos showing droplet shearing from large, submerged hydrocarbon jets into seawater will be presented.

1This work was supported by the Bureau of Safety and Environmental Enforcement of the DOI.

Sunday, November 24, 2019 3:48PM - 5:32PM – Session G16 CFD: Data-driven Methods 4c3 - Mihai Javanovic, USC

3:48PM G16.00001 Towards Generalizable Data-driven Turbulence Model Augmentations. VISHAL SRIVASTAVA, KARTHIK DURAISAMY, University of Michigan, Ann Arbor — Reynolds Averaged Navier Stokes (RANS) models are based on a mix of physical and phenomenological ideas. Once a model structure is fixed, calibration is typically based on a few canonical flows, and as a result, models are often insufficiently accurate in many general applications. Data-driven techniques present the possibility of more accurate models of complex flows, though generalizability and robustness of such models is an open issue and the topic of the work. We address data-augmented turbulence models with a focus on enforcing consistency of the augmentations with the underlying model. Using a data-driven augmentation is consistent with and satisfies underlying physical laws (such as frame invariance) and known relationships (such as preserving the law of the wall in the large Reynolds number limit). These constraints are either imposed directly at the inference step or implicitly enforced by construction. Sample results are presented for equilibrium and non-equilibrium wall-bounded turbulent flows.

4:01PM G16.00002 A data-driven approach to modeling turbulent decay at non-asymptotic Reynolds numbers. MATEUS DIAS RIBEIRO, German Research Center for Artificial Intelligence (DFKI), GAVIN D PORTWOOD, Los Alamos National Laboratory, Los Alamos, PEETAK MITRA, University of Massachusetts, Amherst, TAN MIHN NYUGEN, NVIDIA Corporation, Santa Clara, BALASUBRAMANIA T NADIGA, MICHAEL CHERTKOV, Los Alamos National Laboratory, Los Alamos, ANIMA ANANDKUMAR, NVIDIA Corporation, Santa Clara, DAVID P SCHMIDT, University of Massachusetts, Amherst, NVIDIA TEAM, UMASS TEAM, LANL TEAM, DFKI TEAM — Dynamic modeling of turbulent processes away from asymptotic parameter limits is an active area of turbulence research. This study considers the transient modeling of the kinetic energy dissipation rate, an important component for turbulence closure models like k – c. While asymptotic analysis of the turbulent dissipation process effectively calibrates the model parameters at high and low Reynolds numbers, these calibrations are inaccurate at intermediate Reynolds with strong dependence on large-scale turbulence properties. In intermediate regimes, model tuning via data-driven regression has a leading-order effect on accuracy such that a purely data-driven approach is sensible. Here, we model the kinetic energy dissipation rate in decaying isotropic turbulence using a NeuralODE, a continuous-depth neural network which models continuous-time processes. After training a model using direct numerical simulations (DNSs) over a range of Reynolds numbers and large-scale turbulence initial conditions, we show that a purely data-driven approach to modeling turbulent dynamics via NeuralODEs provides an attractive solution to turbulence closure in non-idealized parameter regimes.
4:14PM G16.00003 A data-driven approach to modeling turbulent flows in an engine environment. PEETAK MITRA, University of Massachusetts Amherst, MATEUS DIAS RIBEIRO, German Research Center for Artificial Intelligence, DAVID SCHMIDT, University of Massachusetts Amherst — We build on work by Zare, Jovanovic, and Georgiou (JFM, vol. 812, 2017) to develop stochastically forced closure models for the mean flow equations of a turbulent channel flow. Given a subset of steady-state velocity correlations for a turbulent channel flow at a friction Reynolds number of 186, we formulate an inverse problem to determine the forcing statistics to the linearized model that provide consistency with DNS. The resulting stochastically forced linearized model is used to drive the mean flow equations in time-dependent simulations. This provides a connection to the mean velocity profile which perturbs the linearized Navier-Stokes dynamics. The feedback connection is implemented using a function that incorporates a two-way interaction between the mean and second-order statistics of the fluctuating velocity field. By analyzing conditions under which this feedback connection converges, we take a step toward the development of new classes of data-driven turbulence closure models.

4:27PM G16.00004 Toward data-driven stochastically forced turbulence closure models. ARMIN ZARE, University of Texas at Dallas, ANUBHAV DWIVEDI, University of Minnesota, MIHAIOLO JOVANOVIC, University of Southern California — We build on work by Zare, Jovanovic, and Georgiou (JFM, vol. 812, 2017) to develop stochastically forced closure models for turbulence environments. The present model combined with corrections to the shear-induced lift model by Tomiyama showed significantly improved results compared to existing models. In conclusion, both the evaluation and validation procedures showed that the present model based on ANN can estimate the bubble size reasonably well in turbulent bubbly flows.

4:40PM G16.00005 A data-driven approach to simulate turbulent bubbly flows using machine learning for modeling bubble size. HOKYO JUNG, YOUNGJAEE KIM, SERIN YOON, GANGWO HA, Dept. of Mechanical Engineering, Sogang University, JIN HO LEE, HYUNGMIN PARK, Dept. of Mechanical and Aerospace Engineering, Seoul National University, DONGJOO KIM, Dept. of Mechanical Engineering, Kumoh National Institute of Technology, JUNGHWOO KIM, Dept. of Mechanical System Design Engineering, Seoul National University of Science and Technology, SEONGWON KANG, Dept. of Mechanical Engineering, Sogang University — We build on work by Zare, Jovanovic, and Georgiou (JFM, vol. 812, 2017) to develop stochastically forced closure models for turbulence environments. The present model combined with corrections to the shear-induced lift model by Tomiyama showed significantly improved results compared to existing models. In conclusion, both the evaluation and validation procedures showed that the present model based on ANN can estimate the bubble size reasonably well in turbulent bubbly flows.

4:53PM G16.00006 Deep Neural Networks for Data-Driven Turbulence Models. ANDREA BECK, DAVID FLAD, CLAUS-DIETER MUNZ, University of Stuttgart — Machine learning methods and in particular deep learning via artificial neural networks have generated significant enthusiasm in the last years. Since these methods can provide approximations to general, non-linear functions by learning from data without a-priori assumptions, they are particularly attractive for the generation of subspace models for multi scale problems. In this presentation, we present a novel data-based approach to turbulence modelling for Large Eddy Simulation by deep learning via artificial neural networks. We first discuss and define the exact closure terms and generate training data from direct numerical simulations of decaying homogeneous isotropic turbulence. We then present the design and training of artificial neural networks based on local convolution filters to predict the underlying unknown non-linear mapping from the coarse grid quantities to the closure terms without a-priori assumptions. All investigated networks are able to generalize from the data and learn approximations. We further show that selecting both the coarse grid primitive variables as well as the coarse grid LES operator as input features significantly improves training results. Finally, we show how to construct a stable and accurate LES model from the learned closure terms.

5:06PM G16.00007 Generalized Non-Linear Eddy Viscosity Models for Data-Assisted Reynolds Stress Closure. BASU PARMAR, University of Colorado Boulder, ERIC PETERS, Ball Aerospace, KENNETH JANSEN, ALIREZA DOOSTAN, JOHN EVANS, University of Colorado Boulder — The prediction of turbulent flow is critical for the design and analysis of engineering systems. Unfortunately, Linear (LEV) and Non-Linear Eddy Viscosity (NLEV) models lack predictive capability in practical flow scenarios involving severe flow separation, secondary flows, and adverse pressure gradients. In this talk, we propose Generalized Non-Linear Eddy Viscosity (GNLEV) models for modeling the Reynolds stress tensor. In these models, we assume the anisotropic part of the Reynolds stress tensor is a function of the mean strain rate tensor, the mean rotation rate tensor, and the mean pressure gradient. This approach can be used to obtain a GNLEV model and thus can be used as a base for a second-order RANS model. For each of these basis functions, the coefficients associated with this expansion themselves are scalar functions of the invariants. The exact form of these coefficients is unknown and hence models must be introduced to obtain complete Reynolds stress closure. In this talk, we use a tensor-based feed forward neural network as a surrogate model to predict these coefficients. Numerical results illustrate the effectiveness of the proposed Reynolds stress closure approach.

5:19PM G16.00008 An S-frame Discrepancy Correction for Data-Driven Reynolds Stress Closure. AVIRAL PRAKASH, University of Colorado, Boulder, ERIC PETERS, Ball Aerospace, RICCARDO BALIN, KENNETH JANSEN, ALIREZA DOOSTAN, JOHN EVANS, University of Colorado, Boulder — Scale-resolving simulations demand large computational resources. Therefore, industry often relies on solving mean flow equations. This averaging leads to an unclosed term known as the Reynolds stress tensor. This closure problem is often addressed using linear eddy viscosity (LEV) models which assume alignment of the anisotropic part of the Reynolds stress tensor and mean strain rate tensor. However, the two tensors do not align for many turbulent flows, including those exhibiting flow separation. In this work, we present a strategy for modeling the discrepancy between the Reynolds stress tensor predicted by an LEV model and the actual Reynolds stress tensor. The strategy relies on learning the discrepancy components in the mean strain rate eigenframe. By intelligently selecting model inputs, we arrive at a model that is both frame and Galilean invariant. We can also ensure energy stability using a simple constraint on the diagonal terms of the discrepancy. To learn a computable model, we employ high fidelity DNS data and neural networks. Numerical results illustrate the effectiveness of our discrepancy modeling strategy. We finally discuss how to use model derived turbulence variables rather than DNS data in the learning process.
In this talk, we will present how to use the TBNN construction to model the turbulent scalar flux in a way that can be readily applied to a RANS framework. When a turbulent flow carries a scalar, such as heat or a contaminant, the turbulent scalar flux (a vector) needs to be modeled concurrently in the RANS framework. Only the prediction of a symmetric, traceless tensor (which is the case for the turbulence anisotropy) is required. In the scenario where a turbulent flow is represented by an invariant vector basis, the tensorial turbulent diffusivity which is predicted from extrapolating while the simulation iterates to convergence. The training and testing sets include the flow over an asymmetric bump and a 180-degree U-bend. When applied to the U-bend, the TBNN-RANS improves predictions of the mean velocity deficit and eliminates unphysical secondary flows predicted by the baseline k-omega RANS model. After quantifying its performance, we compare our formulation to previous algebraic stress models and the optimal basis representation derived by Gatski and Jongen (2000).

1Funding provided by Bosch Corporation

4:01PM G17.00002 Tensor Basis Neural Networks for Turbulent Scalar Flux Modeling

The Tensor Basis Neural Network (TBNN) model developed by Ling et al. (2016) has shown great promise to improve the momentum equations in Reynolds-averaged Navier-Stokes (RANS) solvers. It uses physical insight together with machine learning paradigms to embed rotational invariance into a deep neural network, and then predicts a turbulence anisotropy tensor that obeys this property. The original formulation allows only the prediction of a symmetric, traceless tensor (which is the case for the turbulence anisotropy). In the scenario where a turbulent flow carries a scalar, such as heat or a contaminant, the turbulent scalar flux (a vector) needs to be modeled concurrently in the RANS framework. In this talk, we will present how to use the TBNN construction to model the turbulent scalar flux in a way that can be readily applied to a RANS solver. Manipulation of the appropriate invariant vector basis leads to a form with a general, tensorial turbulent diffusivity which is predicted by the deep neural network at test time. We apply this model to an inclined jet in crossflow and obtain significant improvement in the mean scalar concentration prediction.

1supported by Honeywell Aerospace

4:14PM G17.00003 CFD-ready Turbulence Models from Gene Expression Programming: Concepts

The gene expression programming (GEP) method is applied to develop Reynolds-averaged Navier-Stokes (RANS) models via symbolic regression. The candidate models, represented by strings of genes competing and evolving in the training, can be interpreted as explicitly given equations. Thus, the resulting model, which minimizes the cost function, can be directly implemented into RANS solvers. Based on the advantages of the GEP method, two training strategies have been proposed to develop CFD-ready RANS models. In the first framework called frozen-training, the models are trained to fit high-fidelity Reynolds stress data. In the second approach, called CFD-driven training, the fitness of candidate models is evaluated by running RANS calculations in an integrated way. Both methods have been applied to model development for wake mixing in turbomachines. New models are trained based on a high-pressure turbine simulation data and embedded within a RANS code so that its predictions are agnostic to errors in any baseline RANS solution. A Gaussian closure, while being 400 times cheaper than the high-fidelity simulation.

1This research used resources of the Oak Ridge Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC05-00OR22725.


The existing turbulence closure in URANS, in addition to being a poor representation of the turbulence length scales also accounts for the deterministic shedding scales twice: through the closure and the scale resolution. We propose an alternative non-linear closure, which is built only for the stochastic scales, i.e., devoid of the shedding scales, allowing URANS to resolve the deterministic unsteadiness. The closure is obtained from a novel symbolic regression algorithm: Gene Expression Programming (GEP), which generates a tangible equation for the modelled anisotropy. Using a high-fidelity dataset as reference, the stochastic component of the anisotropy is extracted by triple decomposing the data, which is subsequently used by GEP to produce the new closure. Once obtained, the closure can be used in isolation within URANS as it does not rely on high-fidelity data anymore. The approach is demonstrated using a zero pressure gradient turbulent wake as the reference dataset. The obtained closure was tested on 6 unseen cases, including pressure gradients, and the model significantly outperforms the existing closure, while being 400 times cheaper than the high-fidelity simulation.

1This work was supported by resources provided by the Pawsey Supercomputing Centre with funding from the Australian Government and the Government of Western Australia.
by using deep neural networks, MUHAMMAD IRFAN ZAFAR\textsuperscript{1}, Virginia Tech, JIEQUN HAN\textsuperscript{2}, Princeton University, HENG XIAO\textsuperscript{3}, Virginia Tech — Recent advances in machine learning techniques have enabled researchers to explore data-driven turbulence models as attractive alternatives to traditional algebraic or PDE-based models. However, current data-driven models are all based on local mapping and thus are only applicable to equilibrium turbulence (as with the eddy-viscosity model and algebraic stress models). In this work, we present a PDE-inspired deep neural network architecture based on non-local mapping, which will be used to discover a turbulent constitutive relation from data. Such a network-based representation retains the non-local transport physics embodied in the Reynolds stress transport equations but avoids explicit modeling of individual terms. Furthermore, the neural network is devised to be frame-independent, which is a basic requirement of all constitutive models. Simple illustrative examples are presented to demonstrate the merits of the proposed framework.

\textsuperscript{1}Kevin T. Crofton Department of Aerospace and Ocean Engineering, Virginia Tech, Blacksburg, Virginia 24060, USA; email: zmuhammadfar@vt.edu

\textsuperscript{2}Department of Mathematics, Princeton University, Princeton, New Jersey 08544, USA; email: jiequn@princeton.edu

\textsuperscript{3}Kevin T. Crofton Department of Aerospace and Ocean Engineering, Virginia Tech, Blacksburg, Virginia 24060, USA; email: hengxiao@vt.edu

5:06PM G17.00007 Deep learning based sub-grid scale closure for LES of Kraichnan turbulence\textsuperscript{1} — SIURAJ PAWAR, OMER SAN, School of Mechanical & Aerospace Engineering, Oklahoma State University, Stillwater, Oklahoma - 74078, USA., ADIL RASHEED, Department of Engineering Cybernetics, Norwegian University of Science and Technology, N-7465, Trondheim, Norway. — Performing high-fidelity simulations of large scale multiscale flow problems that resolves fine spatiotemporal features is computationally intractable. Performing eddy simulation (LES) techniques at reducing the computational resources by resolving large scales of the flow and the effect of small scales is modeled. In the present work, we put forth data-driven sub-grid scale closure models for LES of two-dimensional Kraichnan shear stress. The shear stress without averaging in the homogeneous direction(s), which is often adopted for the use of the dynamic Smagorinsky model. The present NN model is applied to a higher Reynolds number ($Re \approx 720$) with the model trained at lower Reynolds number. The results also show good agreements with those of filtered DNS data.

\textsuperscript{1}Supported by NRF-2016R1E1A1A02921549

5:19PM G17.00008 Improving linear embedding of complex nonlinear flow dynamics\textsuperscript{1} — NIKOLAUS ADAMS, LUDGER PAEHLER, Technical University of Munich — We propose an improvement on the concept of linear embedding of nonlinear flow dynamics by a Koopman-mode encoding network. A solution representation of approximate Koopman modes enables a linear estimation of the time evolution on a reduced number of degrees of freedom. Lusch et al., Deep learning for universal linear embeddings of nonlinear dynamics. Nature Communication, 2018, have proposed an encoder-decoder deep learning approach of approximate Koopman projection, and have demonstrated application feasibility for dynamical systems with continuous spectra. The most complex flow considered by Lusch et al. is that of low-Reynolds-number incompressible 2D cylinder flow. The objective of our work is to obtain a better representation of latent dynamics in order to represent significantly more complex flow dynamics. The concept is to improve on the auto-encoding capability of the deep learning approach with a probabilistic objective and by including input information. We demonstrate feasibility of the approach for broadband flow dynamics such as generated by 3D Taylor-Green vortex-flow transition. Also, we will consider the representation of compressibility effects in oscillating gas-bubble dynamics.

\textsuperscript{1}European Research Council (ERC) Advanced Grant 667483
4:01PM G18.00002 Turbulent intensity Enhancement by gaps in submerged Canopy Flows , HAYOON CHUNG, TRACY MANDEL, MARGARITA DRONOV, JEFFREY KOSEFF, Stanford University — Canopies such as seagrass alter their dynamical environment by impacting the flow and turbulent structures. However, unlike well-studied systems in which fully developed boundary layers encounter homogenous and uniform canopies, many aquatic canopy systems display patchiness, e.g., gaps and clearings, that impact the flow, and also interact with velocity fields that are still developing. Therefore, what are the impacts of gaps, both locally and at the canopy scale? We conducted experiments in a recirculating flume with a model vegetation canopy. The canopy is broken up by gaps of varying lengths, and velocities and turbulence observations show the decay of the mixing layer in the gap before it re-enters the downstream canopy segment. The ML decays into a more linear profile, and spreads more shear and turbulent intensities throughout the water column, particularly higher up away from the canopy roughness. Therefore, when compared to an uninterrupted homogenous canopy, the fragmented canopy experiences enhanced turbulent energy at various locations along the canopy. We studied the impact of gap lengths on the extent of turbulence enhancement, and the extent to which the disturbance propagates along the canopy.

4:14PM G18.00003 Influence of plant flexibility on turbulent aquatic canopy flow , SIDA HE, LIAN SHEN, University of Minnesota — Aquatic plant plays an important role in the hydrodynamics of aquatic environments, such as river, lake, and ocean. We have simulated the turbulent flow in submerged aquatic canopy with plant flexibility varying from absolute rigid to extremely flexible. Different from the classical approach of modeling the hydrodynamic effect of canopy as a volume drag force, we resolved the hydrodynamic effect of every plant by an immersed boundary method. Our approach has two benefits: (1) the interaction between the turbulent flow and canopy can be inspected directly; (2) a priori constant drag coefficient is not required. We found that the value of plant flexibility on the turbulent flow and canopy waving motion, i.e., monami. For all canopy flexibility, the velocity profile is self-similar in the mixing layer, though it does not completely follow the hyperbolic tangent profile of pure mixing flow. Our comparison of the mean momentum transport in the flexible and rigid canopy shows the significance of Reynolds stress and dispersive flux in the flexible canopy. We also studied the dispersion relation of monami and found that the wave speed of the high-wavenumber component depends on the plant flexibility.

4:27PM G18.00004 Vibrationally enhanced energy fluctuation in compressible isotropic turbulence with thermal non-equilibrium , QINMIN ZHENG, JIANCHUN WANG, MINPING WANG, SHIYI CHEN, Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology; HUI LI, School of Power and Mechanical Engineering, Wuhan University — The vibrationally enhanced energy fluctuation of compressible isotropic turbulent flows in vibrational non-equilibrium is investigated numerically at turbulent Mach numbers of 0.44 and 1.09, focusing on the effect of the characteristic vibrational relaxation time on statistical features of the dissipation/production components of vibrational energy fluctuation. The dissipation/production of vibrational energy fluctuation might result from effects of dilatation, thermal diffusion and vibrational relaxation. The dissipation component due to thermal diffusion always suppresses the vibrational energy fluctuation in both of compression and expansion regions for the weakly and highly compressible turbulences; but its effect is insignificant comparing to other two components. For the weakly compressible turbulence, the dissipation/production of vibrational energy fluctuation mainly comes from effects of dilatation and vibrational relaxation when the characteristic vibrational relaxation time is small; and the vibrational relaxation component loses its significance gradually with the increase of characteristic vibrational relaxation time. For the highly compressible turbulence, both of the dilatation and vibrational relaxation effects play an important role in the dissipation and production of vibrational energy fluctuation.

4:40PM G18.00005 Enhanced Drag Reduction by High Mach Number Streaming1 , TAIPISH AGARWAL, BENI CUKUREL, IAN JACOBI, Technion — Significant drag reduction for a laminar boundary layer is predicted for high Mach number, travelling-wave flow actuation. Historically, low Mach number, temporal or standing waves were used to modify a base laminar flow by Stokes streaming. Predictions for the associated drag reduction were based on asymptotic, high frequency approximations of the governing momentum balance. We present a numeric solution of the full momentum balance for the streaming flow, based on Lin’s Reynolds decomposition analysis, which allows for examination of the complete range of forcing frequencies. The solution also provides for analysis of travelling wave actuation, which is better suited for practical implementation in boundary layers and channels. The streaming-induced drag reduction is studied as a function of the Mach number, Reynolds number, forcing amplitude, and frequency. In particular, high Mach number, high frequency travelling waves are predicted to produce significant laminar drag reduction. 1United States - Israel Binational Science Foundation Grant Number 2016358

4:53PM G18.00006 Laminar flow in planar Tee joints , MARCOS VERA, Universidad Carlos III de Madrid, GUSTAVO A. PATIO, Instituto Tecnologico Metropolitano, Medellin, IMMACULADA IGLESIAS, Universidad Carlos III de Madrid — We present a numerical investigation of the laminar flow in planar Tee joints, a canonical flow of interest for the thermal-hydraulic design of oil power transformer windings. The steady, constant property flow in planar Tee joints is computed numerically by integrating a non-dimensional formulation of the Navier-Stokes equations with fully developed upstream and downstream boundary conditions. The analysis assumes a straight-through configuration in which the straight duct holds flow in the same direction before and after the junction, whereas the flow from the side branch can combine with the incoming flow or divide from it. We present a description of the flow patterns that emerge in both cases for a full range of mass split ratios, $0 \leq \alpha \leq 1$, several values of straight duct to side branch width ratios, $1 \leq \beta \leq 3$, and Reynolds numbers of the common branch in the range $10^3 \leq \text{Re} \leq 2 \times 10^4$. Flow maps for planar Tee joints are presented, showing the existence of different regions in the $(\text{Re}, \beta)$-plane that exhibit different number and location of recirculation zones. From the pressure distribution, secondary loss coefficients are computed and used to fit pressure loss correlations useful for pipe-network modelling of oil power transformer windings.

5:06PM G18.00007 Considering spatial inhomogeneities in forest canopies , RYAN SCOTT, HAWWA FALIH, SARAH SMITH, NASEEM ALI, JULIAAN BOSSUYT, Portland State University, MARC CALAF, University of Utah, RAUL CAL, Portland State University — Forest canopies serve a key role in determining regional climate and modify terrestrial surface roughness. Forests are typically considered large areas of contiguous roughness elements. While forest composition exists in a state of constant flux, human activities impose an ordered arrangement on local canopy structure. In particular, logging and development produce non-homogeneous patterns of canopy patches separated by regular gaps. To quantify the role of these patterns on the canopy sublayer, a series of wind tunnel experiments were performed with a scaled model forest canopy. The model forest was constructed of multiple patches spanning the width of the wind tunnel. Each patch was comprised of model conifers with a total height of 10 cm and cone diameter of 4 cm. Individual trees were realized by an interconnected reticulated foam canopy layer attached to evenly spaced tree trunks. Particle image velocimetry measurements were collected above the canopy and within gaps for both homogeneous and non-homogeneous canopies. From these measurements, the influence of non-homogeneous canopy patterns is quantified by computing relevant mean statistics and flow structures. In addition, the implications for local effects and land management practices will be discussed in presentation.
from the problem of the typical tilted spectral distribution. Besides, the ANNMM model predicts the PDFs of SGS energy flux much better than the prediction of spectra of velocity and temperature, which almost overlap with the filtered DNS data while the DSM and DMM models suffer from a posteriori almost recovers the average values of the SGS energy flux and SGS energy flux conditioned on the normalized filtered velocity divergence. In an attempt to simulate the turbulent flow over rough walls, the surface roughness is also filtered due to the relatively large grid spacing. The sub-grid-scale roughness can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.

Instead of that, we propose that the effect of surface roughness can be effectively simulated by constraining the near-wall Reynolds stress, which can have a great impact on the turbulent flow, and must be properly modeled. Although some efforts have been made, the LES of turbulent flow over rough walls still remains an open problem. Typically, the roughness effect is modeled by prescribing the instantaneous stress on the first grid point above the wall according to the equilibrium log-law assumption. However this method tends to overestimate the mean stress.
from the Beaufort Gyre Exploration program. Revealing the non-linear resonance within our system. Future work will consider the effects of an external force and incorporate observations.

Our key analysis tool is a novel coordinate rotation, rotating our Boussinesq equations into the space of the waves and so resonance of IGWs. Our work is motivated by the phenomena of layer formation in the upper Arctic Ocean, which prevents the ice packs from the long-lived nature of layers in non-rotating stably-stratified flows and discuss whether our results support the idea that internal gravity waves stratified fluids.

The characteristics of the internal wave field are also altered on varying $\epsilon$ of $R$. The spatial and temporal variability of $\omega$ is systematically varied (by varying $U$), even when normalized with $U_m/D$ (where $U_m = U_c + U_1$ is the maximum barotropic velocity attained by the flow), increases with increasing $R$. The spatial and temporal variability of $\omega$ during the tidal cycle is quantified to further understand the behavior of $\epsilon$. Analysis of vortex dynamics reveal the formation of vortex dipoles during flow reversal, triggered by strong lateral flow and upstream acceleration of fluid in the recirculation zone. The characteristics of the internal wave field are also altered on varying $R$.

This study was supported by National Science Foundation under Grant OCE-1737367.
4:53PM G20.00006 Subsurface Suppression of Turbulence in the Bay of Bengal\(^1\). RAMA GOVINDARAJAN, RITABRATA THAKUR, International Centre for Theoretical Sciences Bengaluru, EMILY L SHROYER, CÉOAS, Oregon State University, J THOMAS FARRAR, ROBERT A WELLER, Woods Hole Oceanographic Institution, JAMES N MOUM, CEOAS, Oregon State University — Quantifying the degree of turbulence and mixing, in an ocean is important to understand how it responds to surface forces and distributes subsurface fluxes. The northern Bay of Bengal is highly salinity-stratified due to the discharge of numerous rivers and precipitation. From a year-long dataset of subsurface turbulent fluxes we show that it is not only surface forcing, but its interplay with an evolving complex subsurface stratification that determines the seasonality in turbulence. We observe a months-long suppression of turbulence below 40m when low-salinity water is present in the topmost layer (Thakur et al. “Seasonality and Buoyancy Suppression of Turbulence in the Bay of Bengal.” Geophysical Research Letters 46.8 (2019): 4346-4355.), in spite of high winds. An implication is that the low mixing with deeper colder water can provide a heat source for tropical cyclogenesis. The stability reasons for this suppression will be discussed.

\(^1\)US Office of Naval Research and Ocean Mixing and Monsoon program of Indian Ministry of Earth Sciences

5:06PM G20.00007 Predicting Interannual Variability of Climate using Deep Learning\(^1\), BALASUBRAMANYA NADIGA, Los Alamos National Lab, CHANGLIN JIANG, AMIR FARIMANI, CMU — Predictability of climate over the interannual to decadal timescale (near term) is controlled by both natural variability related predictability and external-forcing related predictability. Given that the field of near-term prediction of climate is in a nascent stage of development, we examine a deep learning approach to the problem. Preliminary work using a Long Short-Term Memory network architecture with added encoding and decoding is found to be capable of predicting an Earth System Models leading modes of global temperature variability with prediction lead times of up to a year. Related issues and further extensions are discussed.

\(^1\)LDRD program at Los Alamos National Lab

5:19PM G20.00008 Bayesian calibration of global climate models using infrasound events, CHRISTOPHE MILLET, CEA & CMLA, ENS Paris-Saclay, France, FRANCOIS LOTT, LMD, ENS Paris, France, ALVARO DE LA CAMARA, Universidad Complutense de Madrid, Spain — While stochastic parameterizations in Global Climate Models (GCMs) are promising for improving longstanding climate predictions, there is no consensus regarding the values of tunable parameters. In this work, we propose a Bayesian hierarchical approach to calibrate the input parameters of a stochastic multiwave gravity wave (GW) scheme, which is currently in use in the LMD GCM. The GW field is obtained as a combination of individual wavepackets, whose horizontal wavenumber, direction and phase speed are chosen randomly. These parameters are inferred using ground-based infrasound records as tracers of small-scale GW variability. In a sense, the acoustic signals are “back propagated” to adjust the GW sources on a daily basis, using for this a WKB approximation of the Taylor-Goldstein equation to represent the upward-propagating GWs. The method is applied using acoustic signals observed at the Norwegian station in August-September every year. These signals are known to be generated by the well-characterized daily ammunition destruction explosions that occur at the Hukkakero site, in northern Finland. The performance of the method is demonstrated by comparing the updated climatology and variability of the middle atmosphere with the reanalysis provided by the European Centre for Medium-Range Weather Forecasts.

Sunday, November 24, 2019 3:48PM - 5:32PM — Session G21 Drops: Impacts with Liquids II 603 - Omar Matar, Imperial College London

3:48PM G21.00001 Streamlined air-cavities formed by non-superhydrophobic spheres impacting water\(^1\), ADITYA JETLY, IVAN URIEV VAKARELSKI, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — The formation of a stable and streamlined gas cavity following the impact of a sphere on a deep pool, is a recently observed phenomenon\(^2\), which shows motion at near-zero drag. This was shown for both Leidenfrost as well as superhydrophobic spheres. We now extend these results by demonstrating that both metallic and non-metallic spheres, with contact angles between \(30\) and \(120\), can also form stable streamlined cavities, when they are dropped from sufficient height above the pool surface, ranging from \(2\) to \(4\) m. The stable streamlined cavity is attached to the sphere surface just above the equator, instead of being wrapped completely around it. This sphere-with-attached-cavity exhibits slightly narrower shape, but retains the near-zero drag and the free-fall velocity is in compliance with the Bernoulli Law of potential flow\(^2\).\(^1\)Vakarelski et al., Science Advances, 3: e1701558 (2017). \(^2\)Vakarelski*, Jetly & Thoroddsen, Soft Matter, 10.1039/C9SM01025D (2019).

\(^1\)This work was funded by King Abdullah University of Science and Technology (KAUST).

4:01PM G21.00002 How nose curvature affects splashing, JESSE BELDEN, Naval Undersea Warfare Center, MATTHEW JONES, Utah State University, Dept. of Mechanical & Aerospace Engineering, AREN HELLUM, ANTHONY PAOLERO, Naval Undersea Warfare Center, TADD TRUSCOTT, Utah State University, Dept. of Mechanical & Aerospace Engineering — The splash and air cavity formation following the impact of a disk on a free surface are well-documented. Such events are characterized by a sub-surface air cavity and an above surface splash crown that ultimately domes over and seals. If slight curvature is given to the face of the disk, however, these phenomena can change quite dramatically, even resulting in cases that suppress cavity formation. In this talk, we examine the effect of nose curvature on the splash and cavity physics and suggest mechanisms responsible for the observed differences. Furthermore, we directly measure body accelerations in order to estimate instantaneous forces, and relate these measurements to high speed images of the cavity and splash phenomena.
4:14PM G21.00003 Impact of a drop containing a bubble onto a liquid surface, SIQI ZHU, MARIE-JEAN THORAVALE, Xi’an Jiaotong University — We study experimentally the dynamics of a drop containing a bubble falling onto a liquid pool of the same liquid. We first propose an experimental setup to control the formation of this compound drop, controlling its stable formation and the volume of the air bubble. Then we systematically vary the falling height to change the impact velocity. We identify three different regimes by using a dual view high-speed imaging setup to observe the dynamics above and below the pool surface. At low impact velocity, the bubble is pushed into the pool. Above a critical impact velocity, the bubble bursts during the impact, releasing the air bubble before it could be entrapped into the pool. Finally, for higher impact heights, the air drag forces the bursting of the bubble during its falls, before impacting onto the liquid pool. We vary the liquid properties and bubble size to understand these two transitions, and propose some physical explanations.

1International Center for Applied Mechanics (ICAM); State Key Laboratory for Strength and Vibration of Mechanical Structures; Xi’an Jiaotong University (XJTU)
2International Center for Applied Mechanics (ICAM); State Key Laboratory for Strength and Vibration of Mechanical Structures; Xi’an Jiaotong University (XJTU)


1Engineering and Physical Sciences Research Council [EP/P026613/1].

4:40PM G21.00005 Numerical Simulations of Drop Impact on Surfactant-Laden Interfaces1, RICHARD CRASTER, ASSEN BACHTVAROV, LYES KAHOUDJ, CRISTIAN CONSTANTE-AMORES, OMAR MATAR, Imperial College London — The impact of drops on solid and fluid substrates is accompanied by rich phenomena that have been the source of fascination for decades. Recent experimental work (Che and Matar, Langmuir, 33, 43, 12140-12148, 2017) has investigated the effect of surfactants on the phenomenon of “crown” splashing and found that they affect significantly the propagation of capillary waves, the evolution of the crown, and the formation of secondary droplets. In this work, we employ three-dimensional direct numerical simulations to examine drop impacts on thin films in the presence of surfactants. We use a hybrid interface-tracking/level-set method for the interfacial dynamics coupled to a convective-diffusion equation for the surfactant concentration to carry out the computations. We vary different surfactant properties (i.e. diffusion, elasticity, and solubility) to study their effect on the phenomena accompanying the drop impact.

1We acknowledge gratefully the contribution of Drs Damir Juric and Jalel Chergui (both from LIMSI, CNRS, France), and Dr Seungwon Shin (Hongik University, South Korea), in terms of code development, and funding from EPSRC (grant EP/K003976/1), and the Royal Academy of Engineering (Research Chair for OKM).

5:06PM G21.00007 Drop-on-drop Impacts of Complex Liquids: the case of blood1, FUJUN WANG, VANESSA GALLARDO, STEPHEN MICHIELSEN, TIEGANG FANG, North Carolina State University — The interaction of an impacting drop with a sessile drop has received rising attention due to its importance in many applications. Most of the prior work focused on water. In this study, the drop-on-drop impact of blood on a glass surface with varying impact velocities was experimentally investigated. Through a fine adjustment, we obtained a complete regime map, including the bouncing, coalescing, jetting, crown formation, fingering, and film breakup. We quantified the maximum spread diameter (Dm) and the final contact diameter on the surface (Df) after drop-on-drop impact. The measurement of Dm was compared with a model for water and modifications were added for the non-Newtonian effects. For the latter, we found a critical Weber number (We), below which Df will remain the same as the sessile drop. Beyond We, an increase of Df was found to be caused by the collapse of the crown. We finally identified the difference between the fingering and film breakup. The film breakup was generated by the off-center impact during the spreading phase while the fingering breakup took place for the center impact during the collapsing process. The presented cases can well mimic the consecutive impact of blood drops in forensic science.

1We acknowledge the support from National Institute of Justice (Award No. 2018-R2-CX-0033)
The role of humidity and pressure in preserving superhydrophobicity for droplet impact\(^1\), HENRY LAMBLEY, THOMAS SCHUTZIUS, DIMOS POULIKAKOS, ETH Zürich — The design of robust superhydrophobic surfaces for impacting water droplets is typically achieved by creating surface structures with capillary (anti-wetting) pressures greater than that of the incoming droplet (dynamic, water hammer). Recent work has focussed on how competition between compression and drainage of air within the surface texture dictates the evolution of the intervening air layer between droplet and substrate and its role in promoting impalement under ambient conditions through local increases in the droplet curvature. However, little consideration has been given to the influence of the intervening air layer composition, and compressibility effects, when departing from ambient conditions on the impact outcome. Here, we explore the limits of the working envelope for robust superhydrophobic surfaces by varying the ambient pressure and water vapour content. By testing rationally engineered materials with both micro and nanoscale features, we are able to provide additional design constraints and propose solutions for future applications of superhydrophobic and iceophobic technologies across a broad range of environmental conditions.

\(^1\)Partial support of the Swiss National Science Foundation under Grant No. 162565 and the European Research Council under Advanced Grant No. 669908 (INTICE) is acknowledged.

Multilayered synthetic feathers for enhanced underwater superhydrophobicity \(^1\), ZAARA DEAN, FARZAD AHMADI, VIVERJITA UMASHANKAR, Virginia Tech, BRIAN CHANG, Temple University, SUNGHWAN JUNG, Cornell University, JONATHAN BOREYKO, Virginia Tech — Submerged superhydrophobic surfaces can dramatically reduce hydrodynamic drag and bio-fouling, but the enabling air pockets are prone to irreversible collapse. Inspired by ducks, we demonstrate that air pockets within stacked layers of porous superhydrophobic feathers can withstand up to five times more water pressure compared to a single feather. In addition to natural duck feathers, the multilayered effect was replicated with synthetic feathers created by laser cutting micrometric slots into aluminum foil and imparting a superhydrophobic nanostructure. The mechanism for the multilayered enhancement is the more tortuous pathway required for water impalement, which serves to pressurize the enclosed air pockets. This was validated by a probabilistic impalement model and also by filling the feathers with an incompressible oil, rather than air, to suppress the multilayered effect.

Surfactant droplets on hydrophobic microstructures\(^1\), PEICHUN AMY TSAI, NADIA SHAROT, MASOUD B. BIGDELI, JANET ELLIOTT, University of Alberta — Surfactantsamphiphilic molecules can readily adsorb at interfaces. Their presence can destroy the useful, gas-trapping (Cassie-Baxter, CB) wetting state of a drop sitting on a superhydrophobic surface. Here, we examine how surfactants alter the wetting state and contact angle of aqueous drops on hydrophobic microstructures of different surface roughness (r) and solid fraction (\(\phi\)). Experimentally, at low surfactant concentrations (\(C_s\)), some drops attain a homogeneous wetting state (Wenzel, W), while other drops are in the CB state. In contrast, all of our high \(C_s\) drops attain the Wenzel state. To explain this observed transition, we develop a thermodynamic free energy analysis and find that, theoretically, for our surfaces the W state is always thermodynamically preferred, while the CB state is metastable at low C, consistent with experimental results. We further provide a beneficial blueprint for stable CB, gas-trapping states for applications exploiting superhydrophobicity.

\(^1\)NSERC Discovery & Canada Research Chair Program; Alberta Innovates

Dropwise Condensation on Hierarchical Uncoated Metallic Surfaces, DANIEL OREJON, University of Edinburgh, I2CNER, ALEXANDROS ASKOUNIS, University of East Anglia, DANIEL ATTINGER, Iowa State University, YASUYUKI TAKATA, I2CNER, Kyushu University — Dropwise condensation is receiving increasing attention in the past decade. Traditionally, for the fabrication of hydrophilic and superhydrophobic surfaces able to perform in a dropwise condensation manner, the application of a conformal hydrophobic coating was required. In this work, we demonstrate that by passive exposure of our metallic hierarchical micro- and nano-structured copper oxide surface resembling the wetting behavior of a lotus leaf, superhydrophobicity and dropwise condensation can be achieved. The change in wettability from hydrophilic to hydrophobic is due to the adsorption of volatile organic compounds present in the ambient. On contrast, on a similar copper etched surface without copper oxide nanostructures resembling the wetting behavior of a rose petal, filmwise is the final condensation reported. Experimental observations at the micro- and at the macro-scale coupled with droplet morphology and surface coverage analysis, as well as a surface energy analysis are presented to support the different condensation behavior. We conclude on the feasibility of dropwise condensation by the coupling mechanisms of surface structure and hydrocarbon adsorption without the need for a manmade hydrophobic coating. The authors gratefully acknowledge WPI-I2CNER and ThermoSmart for their support.
4:40PM G22.00005 Quantification of Wind-Driven Water Droplets over Surfaces with Different Wettabilities, HUI HU, ZICHERN ZHANG, LIQUN MA, Iowa State University — In the present study, a comprehensive experimental study is performed to quantify the transient runback behavior of water droplet/rivulet flows as driven by boundary layer winds over the surfaces of test plates with different wettabilities. The experimental study is conducted in a low-speed wind tunnel available at Iowa State University (ISU-IRT). A suite of advanced flow diagnostic techniques, which include high-speed photographic imaging, digital image projection (DIP), particle image velocimetry (PIV), are used to quantify the transient runback behavior of water droplets over test plates as driven by the boundary layer winds. Water droplets with their volumes changing from 10 to 100 µL are tested under different incoming wind speeds. In addition to measuring the airflow velocity field around the wind-driven water droplets/rivulets, dynamic shape changes and stumbling runback motion of the water droplets/rivulets are also measured in real time in terms of water film thickness distribution, contact line moving velocity and wet surface area over the test plates with different wettabilities. The findings derived from the present study would be very helpful to gain a better understanding about the important microphysical process, which could lead to improvements of icing accretion models for more accurate prediction of ice formation and accretion process as well as the development of effective anti-/de-icing strategies for aircraft icing mitigation. /abstract- Zichen Zhang, Liqun Ma

4:53PM G22.00006 Out-of-Plane Self-Propulsion of Droplets on Heated Lubricant-Impregnated Surfaces, SUSMITA DASH, Indian Institute of Science, JOLET DE RUITER, Wageningen University, KRIPA VARANASI, Massachusetts Institute of Technology, SUSMITA DASH TEAM, JOLET DE RUITER TEAM, KRIPA VARANASI TEAM — The dynamics of droplets on heated surfaces is crucial for heat transfer applications such as spray cooling. Here we report on the behavior of millimetric water droplets on heated liquid impregnated surfaces (LIS) that are stable at high temperatures. Next to a gentle in-plane hopping motion, droplets can demonstrate one of two vigorous behaviors — at temperatures far below the typical Leidenfrost temperature: either a trapped gas bubble expands to “blow-up” the droplet into a thin liquid shell, or the droplet “jumps” out-of-plane. While the in-plane motion of the droplet is on the order of 10 mm/s, the droplet is propelled vertically upwards to several times its diameter at a velocity of approximately 200 mm/s. We present the mechanics underlying this behavior of droplets, which is specific to lubricant-impregnated surfaces and crucially depends on the thermodynamic state of the impregnating liquid.

5:06PM G22.00007 Impact-induced jets of highly-viscous liquids using a simple syringe1. HAJIME ONUKI, KYOTA KAMAMOTO, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology — In 3D manufacturing, the increment of the executable liquid viscosity is crucial. This study introduces a new method for generating a jet of highly viscous liquid. In our method, a syringe is partially filled with a liquid. The meniscus near the tip of the syringe initially has a concave shape, which induces the flow-focusing effect during the jet formation. The jet emerges when the short-time impact (e.g., collision with the rigid floor) is applied. Remarkably, the highly viscous liquid up to 1,000 cSt is ejected as a liquid jet. The speed of the jet can be controlled by the liquid height inside the syringe. We discuss the reason why the new method can eject the highly viscous liquid based on our previous theory (Onuki et al., Phys. Rev. Applied 2018). We find that, thanks to the geometrical relation, the gradient of the pressure impulse (time integration of the pressure) inside the tip of the syringe is strongly increased, resulting in increasing the jet speed.

1)JSPS KAKENHI Grants (No. 17H01246, and 17J06711)

Sunday, November 24, 2019 3:48PM - 5:32PM –
Session G23 General Fluid Dynamics: Viscous Flows 605 - Neil Balmforth, University of British Columbia

3:48PM G23.00001 Experiments and analysis of viscous flows in bistable elastic channels, OFEK PERETZ, Faculty of Mechanical Engineering, Technion - Israel Institute of Technology, ANAND MISHRA, ROBERT SHEPHERD, Organic Robotics Laboratory, Cornell University, Ithaca, New York, AMIR D. GAT, Faculty of Mechanical Engineering, Technion - Israel Institute of Technology. — COLLABORATION — We present experimental results of viscous fluid propagating into a slender channel with bi-stable cross-section shape, emanating from an upper surface which is a compressed curved elastic sheet. During the propagation of the liquid into the channel, the surface snaps from one stable shape to the other, and a moving front is observed. This front includes wrinkling of the elastic surface, and is shown to provide a stable transition between the two stable shapes of the channel. The viscous flow is analyzed via applying the lubrication approximation and examining self-similarity. For the case of constant inlet pressure, the propagation rate of the transition region is presented for various physical limits. Good agreement is obtained between the experiments and analysis.

4:01PM G23.00002 Hollow-Fiber Microfiltration Systems: Stokes Flow Solution in a Semi-Infinite Channel with Permeable Walls, FRANCESCO BERNARDI, NICHOLAS G. COGAN, M. NICHOLAS J. MOORE, Florida State University — Most wastewater management facilities aimed at water purification in the United States utilize hollow-fiber microfiltration. In these systems, pipes are split into thousands of micrometer-scale capped tubes with permeable walls. As wastewater flows through the filter, foulants are captured by the membraned walls allowing clean water to exit. Understanding the fluid dynamics is a fundamental step towards controlling the fouling process and enhancing the efficiency of microfiltration. We investigate the flow of wastewater through a single hollow-fiber tube. Starting from an infinite channel with permeable walls, we solve the Stokes flow problem in the channel interior for all permeability regimes. Then, we generalize the result to a semi-infinite channel with permeable walls capped at one end to mimic a single hollow-fiber system. Comparison with experiments and future directions will be discussed.

4:14PM G23.00003 Three-dimensional Hiemenz Stagnation-Point Flows, PATRICK WEIDMAN, Department of Mechanical Engineering — A modification of Hiemenz’ two-dimensional outer potential stagnation-point flow of strain rate $\alpha$ is obtained by adding periodic radial and azimuthal velocities of the form $br \sin 2\theta$ and $br \cos 2\theta$, respectively, where $b$ is a shear rate. This leads to the discovery of a new family of three-dimensional viscous stagnation-point flows depending on the shear-to-strain rate ratio $\gamma = b/\alpha$ that exist over the range $-\infty < \gamma < \infty$ with reflectional symmetry about $\gamma = 0$. Numerical solutions for the wall shear stress parameters and the displacement thicknesses are given and compared with their large-$\gamma$ asymptotic Behaviors. Sample similarity profiles are also presented.
E. coli, the form of helices in viscous shear flows migrate across streamlines as a result of their chiral shape and the shear flow. This chirality-induced.TESSER, ESPCI Paris, ANDREAS ZTTL, TU Wien, JUSTINE LAURENT, OLIVIA DU ROURE, ANKE LINDNER, ESPCI Paris — Particles in helical morphologies. Our theory highlights why helical coiling is so ubiquitous in strain-dominated flows.

The origin of helical buckling, we also perform a weakly nonlinear stability analysis. Following a linear Euler buckling regime induced by compressive stresses, unstable planar modes are shown to interact in the presence of geometric nonlinearities and spontaneously give rise to three-dimensional helical morphologies. Our theory highlights why helical coiling is so ubiquitous in strain-dominated flows.

1Leverhulme Trust

4:40PM G23.00005 Helical buckling of flexible filaments in viscous flow. BRATO CHAKRABARTI, University of California, San Diego, YANAN LIU, Northwest University, Xi'an, China. JOHN LAGRONE, RICARDO MARTÍNEZ, LISA FAUCI, Tufts University, OLIVIA DU ROURE, ESPCI, Paris, DAVID SAINTILLAN, University of California, San Diego, ANKE LINDNER, ESPCI, Paris — Helical morphologies of slender structures in flow are generic and have been observed in various experiments and manufacturing processes over a range of length scales. In contrast with the classical helical buckling of elastic rods that requires application of end moments, helical buckling of freely suspended filaments is a spontaneous symmetry breaking induced by distributed viscous forces. In a step towards elucidating this phenomenon, we demonstrate using microfluidic experiments that actin filaments first buckle in compressional flow due to viscous stresses and subsequently form coiled conformations, and two complementary sets of simulations in different geometries also reveal the same. The radius of the emerging helices is found to be independent of filament length, which we explain using a scaling law. To explain the origin of helical buckling, we also perform a weakly nonlinear stability analysis. Following a linear Euler buckling regime induced by compressive stresses, unstable planar modes are shown to interact in the presence of geometric nonlinearities and spontaneously give rise to three-dimensional helical morphologies. Our theory highlights why helical coiling is so ubiquitous in strain-dominated flows.

5:06PM G23.00007 Sedimentation of polygonal tiles. NARAYANAN MENON, ALYSSA CONWAY, Department of Physics, U. of Massachusetts Amherst, RAHUL CHAJSWA, TIFIR International Center for Theoretical Sciences, RINTARO KIRIKAWA, Department of Physics, U. of Massachusetts Amherst, SRIRAM RAMASWAMY, Department of Physics, Indian Institute of Science — We study the stokesian sedimentation of planar shapes by experiments in which polygonal tiles are placed in a vertical plane and sedimented at low Reynolds numbers number in a quasi-two-dimensional container. We focus on the effect of shape-polarity by studying isosceles triangles of varying apex angles. Unlike nonpolar shapes, a triangle rotates as it sediments due to coupling between the orientational and translational degrees of freedom, and asymptotically approaches a stable orientation [Jayaweera, Mason, J. Fluid Mech. 22 (1965)]. For small apex angles the triangle is stable with apex pointing down along the gravity direction. As the apex angle is increased we find a transition at π/3 for which all orientations of the triangle are stable and for apex angles greater than π/3, the triangle is stable with apex pointing up. We understand the experimental results with a model of three stokeslets fixed to the vertices of a triangle. The transition described above, and the coupling of orientation to horizontal drift are captured by this model. We also test generalizations of this discrete-stokeslet model to other regular and irregular polygons and to concave shapes.

N. M. A. C. acknowledges funding through NSF DMR 1507650.

5:19PM G23.00008 Motion of an approximate sphere in a Brinkman medium. D. PALANIAPPAN, Department of Mathematics & Statistics, Texas A&M University, Corpus Christi, O. S. PAK, Department of Mechanical Engineering, Santa Clara University — The motion of an approximate sphere through a porous medium modeled using the Brinkman equation is investigated. Analytic solutions for the velocity and pressure fields due to the translation of a perturbed sphere in a Brinkman medium are found using the Stokes stream function approach. Explicit expression for the stream function is obtained to the first and second order in the small parameter characterizing the deformation. The cases of prolate and oblate spheroids, which depart only slightly from the spherical shape form, are considered as particular examples and the hydrodynamic force on these non-spherical bodies are evaluated. Beyond the first order of deformation, it is found that the hydrodynamic drag on a non-spherical body depends on the permeability coefficient in manners different from the case of a perfect sphere. These differences suggest the complex interactions between non-sphericity and permeability. Several special cases are deduced from our exact solutions. The results may be applied to investigate the effects of particle geometry in transport and locomotion in porous media.
pressible gas-liquid flows, FABIAN DENNER, FABIEN EVRARD, BEREND VAN WACHEM, Otto-von-Guericke University Magdeburg — Simulating compressible gas-liquid flows, e.g. air-water flows, presents considerable numerical challenges due to the stiff pressure-density-temperature relationship of the liquid and the sharp difference in compressibility at the fluid interface. We present a fully-coupled pressure-based algorithm for the simulation of interfacial flows in all Mach number regimes, based on a conservative finite-volume discretisation and a VOF-PLIC method to represent the interface, which treats the continuity equation as an equation for pressure and solves the discrete governing equations in a single linear system of equations. In this contribution, we focus especially on the implementation of the discretised governing equations and on different thermodynamic closures based on the Noble-Abel-stiffened-gas model in fully-compressible and polytropic form. Results of representative test-cases, e.g. pressure-driven bubble collapse or the interaction of shocks with bubbles and drops, are presented to scrutinise the presented algorithm and to highlight the differences caused by the considered thermodynamic closure models.

1Air Force Research Laboratories and Taitech, Inc.

4:14PM G24.00003 Pressure-based algorithm and thermodynamic closure for compressible gas-liquid flows, FABIAN DENNER, FABIEN EVRARD, BEREND VAN WACHEM, Otto-von-Guericke University Magdeburg — Simulating compressible gas-liquid flows, e.g. air-water flows, presents considerable numerical challenges due to the stiff pressure-density-temperature relationship of the liquid and the sharp difference in compressibility at the fluid interface. We present a fully-coupled pressure-based algorithm for the simulation of interfacial flows in all Mach number regimes, based on a conservative finite-volume discretisation and a VOF-PLIC method to represent the interface, which treats the continuity equation as an equation for pressure and solves the discrete governing equations in a single linear system of equations. In this contribution, we focus especially on the implementation of the discretised governing equations and on different thermodynamic closures based on the Noble-Abel-stiffened-gas model in fully-compressible and polytropic form. Results of representative test-cases, e.g. pressure-driven bubble collapse or the interaction of shocks with bubbles and drops, are presented to scrutinise the presented algorithm and to highlight the differences caused by the considered thermodynamic closure models.

4:27PM G24.00004 Simulating Multi-Species Compressible Reactive Flow at Low-Mach Number with a High-Order Fully-Implicit All-Speed Flow Solver, ROBERT NOURGALIEV, MATT MCCLELLAND, Lawrence Livermore National Laboratory, LAWRENCE LIVERMORE NATIONAL LABORATORY COLLABORATION — We present a high-order, fully-implicit all-speed fluid dynamics solver for simulating multi-species compressible reactive flow at very low-Mach numbers. The work is motivated by the development of high-explosive cookoff simulations, which requires modeling multi-species/multi-phase reactive melt convection physics over long time-scales. The governing equations are discretized in space up to 5th-order accuracy with a reconstructed Discontinuous Galerkin method and integrated in time with L-stable fully implicit time discretization schemes. The resulting set of non-linear equations is converged using a robust physics-block based preconditioned Newton-Krylov solver. We demonstrate that our fully-implicit flow solver is able to robustly converge multi-species compressible flow calculations with Mach numbers less than 10^-5. Furthermore, our fully-implicit framework allows for large time steps relative to fast chemical kinetic timescales, which result in highly stiff linear systems.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

4:40PM G24.00005 A Dual Scale Model for Reconstructing Sub-Filter Shear Driven Instabilities, AUSTIN GOODRICH, MARCUS HERRMANN, Arizona State University — A method to compute sub-filter shear-induced velocity on a liquid-gas interface for use in a dual-scale LES-DNS model is presented. The method computes velocity perturbation growth rates by constructing a linear eigenvalue problem based on the well known Orr-Sommerfeld equation using a velocity profile approximated with an error function scaled by the far-field velocities and a prescribed boundary layer thickness. The Orr-Sommerfeld equation, along with appropriate boundary and interface conditions, is then solved numerically with a Chebyshev collocation method as outlined by Schmid and Henningson (2001). The eigenfunctions of the Orr-Sommerfeld equations are expanded into Chebyshev polynomials and evaluated at their Gauss-Lobatto points for spectral accuracy, resulting in an algebraic eigenvalue that can be solved using a standard linear algebra package. With the unstable growth rates computed, the streamfunction definitions are used to compute the normal velocities at the liquid-gas interface. The Chebyshev method is tested under a variety of conditions, and results are presented and compared against prior literature.

1National Science Foundation

4:53PM G24.00006 A stabilized coupled level set and volume of fluid method for incompressible two-phase flow at high Reynolds number and high density ratio, HAN LIU, Department of Mechanical Engineering & St. Anthony Falls Laboratory University of Minnesota, QIANG GAO, Department of Mechanical Engineering University of Minnesota, LIAN SHEN, Department of Mechanical Engineering & St. Anthony Falls Laboratory University of Minnesota — When coupled level set and volume of fluid (CLSVOF) method is used coupling with non-conservative schemes, it can suffer from instability issue when the Reynolds number or the density ratio is high. We present an improved CLSVOF method that is able to simulate two-phase flows at very high Reynolds numbers and large density ratios on Cartesian grid while having high accuracy for turbulent flow resolution. To reduce the error near the two-phase interface, a consistent treatment of mass and momentum transport in the conservative form of discrete equations is employed to solve the nonlinear inertial terms of the Navier-Stokes equations. To resolve the discontinuous momentum across the two-phase interface without oscillation while keeping the accuracy of the numerical solution, a WENO scheme is used for the reconstruction of both velocity and density. The accuracy and robustness of this method are validated by benchmark tests. Quantitative comparisons have been made to show the capability of the method handling realistic two-phase flow problems at high Reynolds numbers and high density ratios.
5:06PM G24.00007 On the use of traction outlet boundary conditions for turbulent multiphase flows1, CYRIL BOZONNET, Grenoble Alpes University, OLIVIER DESJARDINS, Cornell University, GUILLAUME BALARAC, Grenoble Institute of Technology — Due to the finite nature of numerical simulations, computational domains need to be truncated and artificial boundary conditions need to be introduced to close the system of equations being solved. In the context of turbulent multiphase flows, this artificial boundary may lead to the development of unstable backflow patterns in outflow regions and can be the source of wave reflection. Moreover, if the flow is incompressible, the position of the artificial boundary can durably impact the upstream flow. The stabilized traction-free boundary condition has already been introduced in order to mitigate backflow instabilities. In this talk, we will present the improvements that can be obtained by using a non-zero traction boundary condition. Specifically, error level, effect of domain truncation, and surface wave reflection are analyzed. This novel outlet boundary treatment is presented in the context of pressure projection algorithms and interface capturing methods.

1This work was sponsored by the Office of Naval Research (ONR) as part of the Multidisciplinary University Research Initiatives (MURI) Program, under grant number N00014-16-1-2017, and by the IDEX UGA ”International Strategic Partnerships” program. G. B. is also grateful for the support from Institut Universitaire de France.

5:19PM G24.00008 A numerical model for liquid-vapor flows with arbitrary heat and mass transfer relaxation times and general equation of state1, MARICA PELANTI, ENSTA Paris, MARCO DE LORENZO, Compressor Controls Corporation, PHILIPPE LAFOREN, EDF — We describe liquid-vapor flows by a single-velocity six-equation two-phase compressible flow model with relaxation source terms accounting for volume, heat, and mass transfer. The system of equations is numerically solved by a classical fractional step algorithm, where we alternate between the solution of the homogeneous hyperbolic portion of the model system via a HLLC-type finite volume scheme, and the solution of a sequence of systems of ordinary differential equations for the relaxation source terms driving the flow toward mechanical, thermal and chemical equilibrium. For an accurate description of the thermodynamical processes involved in transient liquid-vapor flow problems it is often important to be able to simulate both instantaneous and finite-rate relaxation processes. In this work we present new numerical relaxation procedures to integrate interphase transfer terms with two significant properties: the capability to describe heat and mass transfer processes of arbitrary relaxation time, and the applicability to a general equation of state. We show the effectiveness of the proposed computational model by presenting several numerical tests in one and two dimensions, including simulations of depressurization and blowdown experiments.

1This work was partially supported by the French Government Grant DGA N. 201860071004707501.


3:48PM G25.00001 Reducing ice adhesion via in-situ electrolysis, HENRI-LOUIS GIRARD, SRINIVAS SUBRAMANYAN, DAVID KHANG, YANG SHAO-HORN, KRIPA VARANASI, MIT — The formation of ice on solid substrate is a consequence of numerous processes such as loss of shape of airfoils, weight increase of structures, heat transfer through roofs, slipperiness of surfaces, and more. Current de-icing technologies such as resistive heating, mechanical fracturing, or chemical spraying are either energy intensive or environmentally problematic. The use of super-hydrophobic surfaces to mitigate ice adhesion has been explored and shown to lead to significant reductions in the adhesion strength of ice as long as the texture was not penetrated by the ice. Unfortunately, cold surfaces exposed to humid environments are subject to frost formation which fills the defects and reduces the performance of these substrates. Here, we demonstrate the use of water electrolysis as a means to create stress-concentrators within the ice-adhesion interface in situ. This study investigates the capture of bubbles at the ice-solid interface and the effect of these bubbles on the adhesion of ice on the surface.

4:01PM G25.00002 Bubble Breathing during Dissolution in Confined Geometries under Partial Wetting, KE XU, Massachusetts Institute of Technology, AMIR PAHLAVAN, Princeton University, RUBEN JUANES, Massachusetts Institute of Technology — One expects that, during the dissolution of a gas bubble into surrounding liquid, the bubble volume will decrease monotonically as a result of continuous mass loss. This intuitive picture, however, changes in a surprising way when the dissolving bubble is confined in a geometry under partial wetting to the gas and the liquid. We show that the bubble then experiences cycles of volume expansion (“inhaling”) and shrinkage (“exhaling”), until it fully dissolves. This bubble “breathing” is the result of condensation, growth and merging of liquid droplets on the gas-solid surface, and their eventual expulsion from the gas bubble. Theoretical analysis shows that these counter-intuitive dissolution dynamics are driven by a reduction in the system’s free energy. We provide a scaling argument that identifies the transition from this intermittent expansion-shrinkage “breathing” regime and the classical shrinkage-dominated regime. This behavior could play a major role in determining the macroscopic mass-transfer dynamics in partially wetting systems under confinement, such as oxygen transfer in low-temperature fuel cells and enhanced hydrocarbon recovery in porous rocks.

4:14PM G25.00003 ABSTRACT WITHDRAWN –

4:27PM G25.00004 ABSTRACT WITHDRAWN –

4:40PM G25.00005 Bubble under field: similarities and differences, SEBASTIEN MAWET, STEPHANE DORBOLO, HERVE CAPS, University of Liège, CYPRIEN GAY, FLORENCE ELIAS, University of Paris Diderot — Deformations undergone by a droplet submitted to an electrical force have been recently studied by Beroz et al. They propose a law based on the equilibrium between capillarity and electrical force. On the other hand, Ikala et al. focus on the dynamics of a ferrofluids droplet resting on a superhydrophobic surfaces under a a magnetic field. They observe the division of a droplet in two smaller ones. Both works show the current interest for the manipulation and the deformation of droplet. Here, we propose to compare with bubbles. We propose to characterize the shape of the bubble submitted to a electric and a magnetic field. Both applied fields lead to similar deformations of the bubble. In particular, we characterize the shape variations of the surface of the hemispherical bubble immersed in the uniform electric field of a plan capacitor. We also observe those variations for a ferrofluids hemispherical bubble under a uniform magnetic field.
4:53PM G25.00006 Numerical Simulations of Gas-Liquid Dispersion in Millichannels Through Parallel Contacting Sections, XIAOJIN TANG, THOMAS ABADIE, OMAR MATAR, Imperial College London — The generation of a regular train of bubbles or droplets in microchannels has been a challenge for the past decade for optimising microreactor design and operation. Although many contacting methods (e.g. T-junctions, cross-junctions) have been studied for single channels, the effects of operating conditions for generating well-controlled bubble dispersions with multiple contacting sections still requires further understanding. For industrial applications, millichannels can be more useful than microchannels to avoid blockage and for higher throughput. However, few research results in millichannels could be found in the literature. We study the gas dispersion in a liquid flowing in a rectangular microchannel through three parallel contacting sections. A finite-volume solver with a Volume-Of-Fluid method to capture the interface is used. The effects of geometry and operating conditions on the gas flow through the contacting sections are analysed along with the breakup frequencies. The effects of contacting section geometry on bubble size distributions and velocities under which stable bubble trains can be formed are discussed as a step towards developing an optimal structure of millichannels where uniform and highly dispersed bubble swarms can be achieved.

1RAEng/PETRONAS Research Chair for OKM, and SINOPEC

5:19PM G25.00008 Bubble pinch-off in turbulence: shape oscillations and escaping self-similarity, DANIEL RUTH, WOUTER MOSTERT, STPHANE PERRARD, LUC DEIKE, Princeton University — Though bubble pinch-off is an archetype of a dynamical system evolving towards a singularity, it has always been described in idealized theoretical and experimental conditions. Using experiments, simulations, and analytical modeling, we consider bubble pinch-off in a turbulent flow, representative of natural conditions in the presence of strong and random perturbations. We show that the turbulence sets the initial conditions for pinch-off, but once the pinch-off starts, the turbulent time at the neck scale becomes much slower than the pinching dynamics: the turbulence freezes. We show that the average neck size, d, can be described by d ∼ (t − t₀)⁻α, where t₀ is the pinch-off, or singularity time, and α ≈ 0.5, in close agreement with the axisymmetric theory with zero initial flow. Neck shape oscillations set by the initial conditions are described by a quasi-two-dimensional linear perturbation model, and persistent asymmetries in the neck are related to the complex flow field induced by the deformed bubble shape. In many cases, a three-dimensional kink-like structure forms on part of the neck just before pinch-off, causing d to escape its self-similar decrease.

1This work was supported by NSF CAREER award CBET 1844932 and American Chemical Society Petroleum Research Fund 59697-DN19 to L.D.


3:48PM G26.00001 Vorticity Generation in a Single Nanosecond Spark Discharge Due to Shock Curvature, BHAVINI SINGH, LALIT RAJENDRAN, PAVLOS VLACHOS, SALLY BANE, Purdue University — Spark plasma discharges are generated by raising the voltage difference between two electrodes, until breakdown voltage is reached, resulting in ionization of gas in the electrode gap. This rapid release of energy results in the formation of a shock wave as well as a region of hot gas that expands and cools with time. At later times, vortex rings are formed near each electrode that entrain ambient gas into the electrode gap to cool the hot gas kernel. However, the mechanism(s) responsible for the generation of vorticity in the flow field, and the effect of electrical energy deposited in the gap on this vorticity is unclear. We hypothesize that the shock wave formed at the time of the discharge generates the vorticity by means of baroclinic effects due to shock curvature, and this vorticity field then develops into the pair of vortex rings observed at later times. In this work we develop a detailed analytical framework to relate the vorticity generation to the shock curvature and energy deposited in the electrode gap, and test this framework using 700 kHz shadowgraph and 100 kHz pulse-burst Particle Image Velocimetry (PIV) measurements. We extract the shock curvature from the shadowgraph images and compare the predicted vorticity field from the framework with the measured vorticity field from PIV. These measurements along with the framework will help ascertain the role of shock curvature and energy deposited on the vorticity generation.

4:01PM G26.00002 Design of a zero-net-mass-flux actuator based on a DBD jet in a partially enclosed cavity, ANOOD ALKATHEERI, ABDUL RAOUF TAJIK, ABDULLA ALJABERI, VLADIMIR PAREZANOVIC, Khalifa University, UAE — Recently, Lucas et al. [1] have shown that a shallow cavity at the base of a 3D bluff body can significantly stabilize the symmetry breaking mode of its wake. A natural recirculation of the flow near the base opposes the selection of an asymmetric state, which symmetrizes the wake and yields a higher base pressure (reduced drag). Our work investigates the possibility of recreating this effect, using a partially enclosed cavity with a Dielectric Barrier Discharge (DBD) jet inside. When the DBD jet is activated it produces suction and blowing action at the twin slots and can therefore shift the selection of the transition mode. Current work focuses on the optimal design of such a cavity, using 2D URANS-based simulations in conjunction with the electrodynamic force model of a DBD jet [2]. The dynamics of the DBD jet inside a several different cavity shapes are simulated for steady and periodically pulsed actuation, and the velocity profiles are analyzed. The goal is to establish the most important geometric properties of the cavity for a desired balance between suction and blowing action from the two slits using a single DBD jet.

4:14PM G26.00003 On The Physical Mechanism of Turbulent Boundary Layer Drag Reduction Under AC-DBD Plasma Actuation. SAMARESH MIDYA, ALAN DUONG, THOMAS CORKE, FLINT THOMAS, University of Notre Dame — The results of a series of experiments are reported which use near-wall active flow control designed to intervene in the process of streamwise vortex (SWV) generation, which is primarily responsible for turbulence production in wall-bounded flows. The flow control method utilizes an array of flush mounted AC-DBD plasma actuators in a ZPG TBL over the range of Re = 550-1750. The control flow consists of a series of near-wall, span-wise oriented unidirectional wall jets with velocity comparable to the friction velocity and has been shown to produce significant reductions (around 20%) in drag. The control flow is fully characterized using PIV. The span-wise wall jets inhibit the formation of near-wall SWVs and thus reduce the turbulence production. This manifests itself in the reduction of near wall turbulent Reynolds stress producing events. The focus of the reported experiments is to further clarify the mechanism of drag reduction. X-wire measurements utilizing the quadrant splitting technique are performed downstream of the actuator. These are used to characterize & contrast both the duration of & time interval between quadrant 2 & 4 events in the actuated & non-actuated flows. The quadrant contributions to the Reynolds stress are compared for natural & actuated cases. Effort has been made to correlate the observed drag reduction & the change in Reynolds stress profile. The turbulence statistics have also been compared to similar statistics obtained from a ZPG TBL under pulsed-DC plasma actuation where even higher drag reduction was achieved.

4:27PM G26.00004 On The Physical Mechanism of Turbulent Boundary Layer Drag Reduction Under AC-DBD Plasma Actuation. SAMARESH MIDYA, ALAN DUONG, THOMAS CORKE, FLINT THOMAS, University of Notre Dame — The results of a series of experiments are reported which use near-wall active flow control designed to intervene in the process of streamwise vortex (SWV) generation, which is primarily responsible for turbulence production in wall-bounded flows. The flow control method utilizes an array of flush mounted AC-DBD plasma actuators in a ZPG TBL over the range of Re = 550-1750. The control flow consists of a series of near-wall, span-wise oriented unidirectional wall jets with velocity comparable to the friction velocity and has been shown to produce significant reductions (around 20%) in drag. The control flow is fully characterized using PIV. The span-wise wall jets inhibit the formation of near-wall SWVs and thus reduce the turbulence production. This manifests itself in the reduction of near wall turbulent Reynolds stress producing events. The focus of the reported experiments is to further clarify the mechanism of drag reduction. X-wire measurements utilizing the quadrant splitting technique are performed downstream of the actuator. These are used to characterize & contrast both the duration of & time interval between quadrant 2 & 4 events in the actuated & non-actuated flows. The quadrant contributions to the Reynolds stress are compared for natural & actuated cases. Effort has been made to correlate the observed drag reduction & the change in Reynolds stress profile. The turbulence statistics have also been compared to similar statistics obtained from a ZPG TBL under pulsed-DC plasma actuation where even higher drag reduction was achieved.

4:40PM G26.00005 High-speed Schlieren visualizations of plasma pulsed jet in subsonic and supersonic regimes. NICOLAS BENARD, Université de Poitiers, YANG ZHANG, Florida State University, HAO-HUA ZONG, MARIOS KOTSONIS, Delft University of Technology, LOU CATTAFESTA, Florida State University, ERIC MOREAU, Université de Poitiers — A novel type of pulsed jet using a spark discharge has been developed in recent years at PPRIME Institute. The device combines a cavity continuously fed by an external pressure source and a plasma discharge propagating from a small needle aligned with the jet orifice. The electric circuit uses a pre-ionization wave to trigger the ignition of the spark discharge. This system has been developed for flow control applications where high-frequency forcing and high momentum injection are both required. The cavity is supplied by constant air pressure at 2.4 bar, producing supersonic jet from the 1-mm diameter orifice. Subsonic conditions can also be achieved by adding a neck extension including a sudden expansion from 1 to 2 mm. In the present investigation high-speed schlieren visualizations have been conducted at a repetition rate of 100 kHz for both supersonic and subsonic operating modes. The energy released is 100 mJ/pulse, and the visualizations clearly demonstrate the strong modulation of the flow conditions from the jet orifice to the surrounding flow region. Precursor and secondary shock waves are visualized as well as the structure of the pulsed jet with a vortex ring formed in front of the jet.

4:53PM G26.00006 Ac-DBD vs Ns-DBD Plasma Actuation on a Turbulent Mixing Layer. ASHISH SINGH, JESSE LITTLE, University of Arizona — A parametric study is undertaken to compare ac-DBD (momentum) and ns-DBD (thermal) plasma actuators in a low speed turbulent mixing layer using an identical load. The mean flow response to each actuation technique is matched at the imposed equivalence in local control authority between the two actuators extends to the global flow, both in the mean and fluctuating components. The ns-DBD plasma actuator requires six times more energy to achieve the same control as the ac-DBD in this specific flow. By studying the flow field very near the splitter plate trailing edge, the difference in momentum versus thermal actuation mechanisms is revealed. A velocity deficit is observed for both actuators, but a thermal bump-like mechanism is responsible in the ns-DBD case while a near surface jet redirecting momentum is found in the ac-DBD case.

5:06PM G26.00007 Vortex Development in a Laminar Separation Bubble measured via Tomographic Particle Image Velocimetry. JOHN KURELEK, SERHIY YARUSEVYCH, University of Waterloo, MARIOS KOTSONIS, Delft University of Technology — The development of shear layer vortices in a laminar separation bubble is investigated experimentally using Planar and Tomographic Particle Image Velocimetry. The experiments are carried out in a series of wind tunnel tests, with the bubble formed on a flat plate subjected to an adverse pressure gradient. Sensitivity to spanwise uniform (2D) and small-amplitude spanwise modulated disturbances (3D) is explored, with disturbances produced using surface mounted dielectric barrier discharge plasma actuators. Compared to the natural case, both types of forcing lead to earlier vortex formation that is rendered essentially two-dimensional at roll-up. While the vortex filaments remain largely two-dimensional until breakdown when subjected to the 2D forcing, deformations rapidly develop within the separation bubble at the spanwise wavelength that matches the input wavelength when 3D forcing is applied. The results elucidate the mechanism responsible for the observed rapid vortex formations from the initially weak spanwise component of the input disturbances and the associated impact on the mean bubble characteristics.

1The authors gratefully acknowledge the National Sciences and Engineering Research Council of Canada (NSERC) for funding this work.
In the present study, the aerodynamic forces acting on a hovering hawkmoth which is feeding on an artificial flower are quantified. The high-speed Schlieren photography results, on the other hand, provide us clear guidelines about how to setup the experiment and when/where to expect the starting vortex. The flow field quantification from the conventional PIV provides us reliable vortex strength data while the high-speed Schlieren photography experiments provide us critical spatial information about the three-dimensional structure of the vortex loop. With those two temporal data quantified accurately, the instantaneous aerodynamic forces can be estimated in the end.

 Recently strategies for mosquito population control have been largely relying on the control of mosquito population. The flight dynamics of mosquito are closely related to their mating and courtship behavior, therefore understand the fluid dynamics can play an important role in developing novel strategies for mosquito population control. In this work, for the the first time, we measure the wake flow of a single tethered mosquito using time-resolved tomographic particle image velocimetry system. Mosquitos are tethered inside a transparent box, and fog particles are used to seed the area. Four cameras, synchronized with the illumination of a Nd: YLF laser, are employed to record the flow field at a speed of 700 Hz (frame straddling mode). The 4D flow structure, pressure field, and flow statistics at the wake of the mosquito are reconstructed and reported in this study.

# Session G27 Biological Fluid Dynamics: Insect Flight II

**3:48PM G27.00001 Dragonfly’s Righting Reflex**

Z. JANE WANG, JAMES MELFI JR, Cornell University, ANTHONY LEONARDO, Janelia Research Campus, HHMI — Insects must right themselves in air so as not to fall. Exactly how insects manage to right themselves is a large mystery in neural behavior of animals. The goal of our work is to find clues about their internal actions through careful measurements and analyses of their aerial acrobatics, in the case of a dragonfly. A dragonfly falling upside down can right itself in about 200ms. During this brief episode, not only it has to sense its perilous condition, it also has to respond with proper muscle actions so to modulate the flapping wing motions such to that generate enough aerodynamic torque in order to correct its orientation. Here, we measure the intricate wing modulations of all four wings that a dragonfly employs to make such a maneuver. We further develop a computational model to simulate the righting maneuver so to tease out the key wing asymmetry that leads to a successful recovery. By analyzing the falling trajectory, we calculate the muscle torque dragonfly used to drive the body rotation. We further conjecture a sensory motor pathway during the dragonflys righting reflex in response to the perceived horizon.

**4:01PM G27.00002 Measurement of mosquito wake flow using time-resolved tomographic particle image velocimetry**

ZHONGWANG DOU, PAVLOS VLACHOS, Purdue University — Recent strategies on the control of mosquito-borne diseases have been largely relying on the control of mosquito population. The flight dynamics of mosquito are closely related to their mating and courtship behavior, therefore understand the fluid dynamics can play an important role in developing novel strategies for mosquito population control. In this work, for the the first time, we measure the wake flow of a single tethered mosquito using time-resolved tomographic particle image velocimetry system. Mosquitos are tethered inside a transparent box, and fog particles are used to seed the area. Four cameras, synchronized with the illumination of a Nd: YLF laser, are employed to record the flow field at a speed of 700 Hz (frame straddling mode). The 4D flow structure, pressure field, and flow statistics at the wake of the mosquito are reconstructed and reported in this study.

**4:14PM G27.00003 Measurement of starting vortex strength of a hovering hawkmoth**

YUN LIU, Purdue University Northwest — From the high-speed Schlieren photography, salient flow structures were successfully visualized and captured on a near hovering hawkmoth Manduca. During the down-stroke, a vortex loop was formed with evolving size and strength. Originally, optical flow method was implemented on the Schlieren images and vortical flow field was quantified at certain time instants. However, owing to the three-dimensional nature of the flow structure and its corresponding complexities, the vortical flow field quantification is not always successful and significant uncertainties in starting vortex strength estimation exists. Therefore, in this work, a further step is taken for accurately measuring the strength of starting vortex. To achieve this goal, a time resolved 2D-PIV is conducted on a hovering hawkmoth which is feeding on an artificial flower. The high-speed Schlieren photography results, on the other hand, provide us clear guidelines about how to setup the experiment and when/where to expect the starting vortex. The flow field quantification from the conventional PIV provides us reliable vortex strength data while the high-speed Schlieren photography experiments provide us critical spatial information about the three-dimensional structure of the vortex loop. With those two temporal data quantified accurately, the instantaneous aerodynamic forces can be estimated in the end.

**4:27PM G27.00004 New measure based solely on wake data for hovering performance**

SACHIN SHINDE, Indian Institute of Technology, Kanpur, India, JAYWANT ARAKERI, Indian Institute of Science, Bangalore, India — In case of hovering wherein forward speed (thus work) is zero, the conventional definition of efficiency does not hold; Figure of Merit (FM) is an alternative. However, both efficiency and FM require measurement of input power which is not trivial for hovering birds and insects. We propose a new measure, independent of measuring input power, called ‘jet effectiveness factor (Π)’ defined based on mass, momentum and kinetic energy flux data, essentially requiring only wake velocity field which can be gathered using PIV, MTV. Inverse of Π can be thought as efficiency. We theoretically calculate Π for two-dimensional (2D) rectangular, triangular and Gaussian jets as 1, 1.250 and 1.1548 respectively. Closer the value of Π to 1, more effective would be the jet. From PIV experiments, we calculate Π for 2D reverse Karman jet generated by flexible foil flapping in an otherwise quiescent ambient - situation relevant to hovering, and show that the value of Π is close to that for Gaussian jet for several cases obtained by varying pitching amplitude and frequency. This new measure can be extended to calculate effectiveness of jet generated by MAVs, AUVs and cruising fish, birds, insects where measuring input power is non-trivial.

**4:40PM G27.00005 ABSTRACT WITHDRAWN —**
4:53PM G27.00006 Why are long sequences of steady flight less common at higher speeds of forward flight in Hawkmoth?1. CHENG-YU LI, MICHAEL CORBI, Villanova University, TYSON HEDRICK, University of North Carolina at Chapel Hill — The hawkmoth is able to sustain a steady hover or level flight at lower speeds (0.7 m/s). However, previous wind tunnel experiments suggested that long sequences of steady forward flight were less common at higher flying speeds (>2.0 m/s) despite changes to the flight posture and muscle recruitment. Considering hummingbirds have about the same body size and can easily achieve ~10 m/s forward flight speed, it is unclear why hawkmoths were not observed achieving steady fast flight. In this work, high-speed photogrammetry and 3D surface reconstruction were used to investigate a hawkmoth’s (Manduca sexta) wing kinematics at a forward flight speed of 4 m/s. The associated fluid dynamics and wing aerodynamic performance were then studied using an in-house computational fluid dynamics solver. Quantitative analysis has shown a significant amount of negative lift was generated during upstrokes at this high forward flying speed. Such a negative lift in the upstroke might reduce maximum sustained flight speeds in this species and might represent an adaptation for the hovering feeding mode for which this species is known.

5:06PM G27.00007 Effects of wing-wake interactions on the aerodynamic performances of a hovering rhinoceros beetle.1. SEHYEONG OH, BOOGEON LEE, HYUNMIN PARK, HAECHOEN CHOI, Seoul National University — We investigate the aerodynamic performance of a hovering rhinoceros beetle using numerical simulation and a quasi-steady aerodynamic model. The simulation shows that the wing-wake interactions significantly affect the aerodynamic performance. To examine the wake characteristics behind the wing, we obtain the temporal and spatial distributions of downwash motion from numerical simulation, and show that the downwash motion is non-uniform along the wing spanwise direction, of which magnitude is large immediately after the stroke reversal and small at the end of half stroke. Therefore, we model the wake behind the wing as a non-uniform (in spanwise direction) and sawtooth-type (in time) downwash motion. Also, we combine a quasi-steady aerodynamic model and a momentum theory, together with unsteady and non-uniform downwash motion. The aerodynamic performance predicted by the present aerodynamic model is in good agreement with that of the present numerical simulation.

5:19PM G27.00008 Generation of Sound from Flapping Wings of Mosquitoes1. JUNG HEE SEO, Johns Hopkins University, TYSON HEDRICK, University of North Carolina Chapel Hill, RAJAT MITTAL, Johns Hopkins University — It has long been recognized that mosquitoes use “wing tones” (sound generated by flapping wings) for communication and signaling during mating. Previous experimental studies showed that the wing tone based “communication” between conspecifics is complex, and it conveys important information about fitness and sexual interest. The mechanisms of wing tone generation and the characteristics of its sound field are however not well understood. Mosquitoes (e.g. Culex) that employ wing-tone communication have very unique wing kinematics (high frequency and small stroke amplitude) compared to other insects of similar size and it has been speculated that this facilitated wind-tone communication. In the present study, the generation of wing-tones in a mosquito (Culex) is computationally investigated to explore the speculations. The flow field around a hovering mosquito is simulated by solving the incompressible Navier-Stokes equations using a sharp-interface immersed boundary method, and the aeroacoustic sound is predicted by the Ffowcs Williams and Hawkings equation. The analysis of the simulation data suggests that the kinematics employed by these mosquitoes facilitates efficient generation of wing-tones.

1Supported by the Human Frontier Science Program (HFSP).

Sunday, November 24, 2019 3:48PM - 5:32PM – Session G28 Bubbly Flow I 610 - Alison Hoxie, University of Minnesota

3:48PM G28.00001 Bubble induced mixing in stratified fluids within a confined domain1. MAATHANGI GANESH, SANG KYU KIM, SADEGH DABIRI, Purdue University — A numerical study of mixing in temperature stratified fluids induced by the motion of monodispersed swarm of bubbles in a thin gap is carried out. The confinement prevents turbulent production and mixing occurs primarily due to transport by the bubble wake. The confinement also causes the bubbles to be flattened between the solid walls, with a gap to diameter ratio of 0.31. Bubbles entrain liquid from the lower, high-density layers and release it in the low density regions, thus causing mixing between the different liquid layers. Simulations are run for void fractions between 3.35% and 13.4%. The strength of stratification is varied by changing the Froude number between 4.5 and 12.74. We observe a zigzag motion of the bubbles attributed to the periodic vortex shedding behind the bubbles, quantified by the bubble velocity autocorrelations. We report the formation of horizontal clusters, the sizes of which are quantified by the cluster size index, and establish a qualitative correlation between the size of the clusters and the rise velocity of the bubbles. The mixing is characterized by three parameters namely, the COX number, diapycnal eddy diffusivity and mixing efficiency. We report an increase in the buoyancy flux across the isopycnals as void fraction increases. The fraction of energy production due to the buoyancy flux increases with the strength of stratification, giving rise to a higher mixing efficiency.

1Acknowledgement: Supported by NSF Grant No. CBET-1705571

4:01PM G28.00002 Explaining Hydrodynamics- mass transfer interplay in a bubble column column using bubble size distribution1. MANAS MANOHAR MANDALAHALLI, JOHAN NIENHUIS, LUIS MANUEL PORTELA, ROBERT FRANK MUDEDE, Delft University of Technology — Gas-liquid mass transfer in bubbly flows is strongly coupled with its hydrodynamics, an interdependency relevant for many industrial applications. Shrinking bubbles and/or presence of dissolved contaminants influence both transport processes and their interrelation; this makes understanding the interplay quite challenging. In our experimental work, we study the influence of dissolved electrolytes (up to 1M NaCl) on CO2 mass transfer in a homogeneous rectangular bubble column, up to 7% gas fraction. Bubble size distribution, rise velocity and gas fractions are measured by high-speed imaging and digital image processing, while the liquid CO2 concentration by monitoring pH variation. Coalescence inhibition, due to the electrolyte, and dissolving CO2 both lead to a bimodal bubble size distribution with a lower mean bubble diameter, when compared to an ideal N2-water system. Besides an increment in the interfacial area for the mass transfer, the bimodal distribution leads to bubble plume oscillations and recirculation zones in the column; dynamics of the shrinking bubble plume further influences mixing in the column. Our results strongly indicate that the mass transfer can be explained by the influence of the bubble size distribution on the hydrodynamics.

1This work is part of the Industrial Partnership Programme i36 - Dense bubbly flows of the Netherlands Organisation for Scientific Research (NWO)
4:14PM G28.00003 Shaping stress fields with spreading bubble streams: applications for biofouling prevention 1, JAMES BIRD 2, MARK MENESSES, C. FREDERIK BRASZ, Boston University, JESSE BELDEN, Naval Undersea Warfare Center — Hard fouling organisms require significant shear stresses to be removed from a ship hull. Recently it has been shown that more frequent grooming requires less shear stress to remove these fouling organisms. By introducing a stream of air bubbles beneath a submerged surface, one can prevent marine biofouling through a continuously applied average shear stress of approximately 0.01 Pa; however, recent studies have shown maximum shear stresses applied by a bubble can be on the order of 200 Pa. Additionally, it has been shown that the period between critical shear stress events rather than an average stress value may be significant in determining the recruitment of settling marine organisms. Considering these recent developments, we investigate the effect of varying aeration flow rate on the antifouling ability of bubble streams. We find that by decreasing the frequency of bubble production the extent of biofouling prevention also decreases. Using a combination of field data, multiphase PIV, and integral plume theory, we find the shape of the areas influenced by the bubble plumes is related to the lateral spreading of bubbles as they rise.

1We acknowledge support from ONR and NSF GRFP
2There has been confusion in the past regarding another James Bird in our community. This is the only talk in which James C. Bird from Boston University is the presenter.

4:27PM G28.00004 Interaction of a vortex ring with single and multiple air bubbles, SUBHAJIT BISWAS, RAGHURAMAN N GOVARDHAN, Indian Institute of Science — Bubbly turbulent flows occur in many places such as in chemical reactors and in geophysical applications, besides interest from the drag reduction perspective using injected bubbles in a (water) boundary layer. In these flows, the bubbles interact with vortical structures in the flow, this being a two-way coupled interaction. Motivated by such complex interactions of bubbles with vortical structures in turbulent flows, we experimentally study an idealization of this, namely, the interaction of a vortex ring (in water) with single and multiple air bubbles. The main parameters in this case are the ring strength, and a ratio of the bubble volume to ring core volume. In these interactions, we are interested in both the vorticity dynamics and the bubble dynamics, and these are measured using time-resolved PIV and high speed shadowgraphy, respectively. We start with the interaction of a single bubble with a single vortex ring, followed by studies with two or many bubbles (bubble swarm), the latter being closer to the case of bubbly flows. The similarities and differences from the vorticity and bubble dynamics perspectives for the different cases will be presented at the conference.

4:40PM G28.00005 Direct numerical simulation on the effect of slip velocity in bubbly plumes 1, HYUNDUK SEO, Pusan National University, SIVARAMAKRISHNAN BALACHANDAR, University of Florida, KYUNG CHUN KIM, Pusan National University — Bubbly plume is one of the examples of multi-phase flow which is not confined by the sidewall. The bubbly plume has been applied to enhance the mixing for aeration or reaction between the gas and liquid phases. Air-water plume has higher slip velocity rather than other combinations of fluids. In this study, two-phase fluids governing equations are solved by DNS using the spectral element method code Nek5000 with Eulerian-Eulerian approach. To investigate the role of slip velocity in forming a plume structure, we calculated higher-order statistics such as turbulence kinetic energy budget. Not only higher-order statistics but also plume parameters in the conventional integral framework were calculated to quantify and characterize the plume structure. From the statistics, higher slip velocity results in a sharp interface between the inner core region and the outer annular region of the plume. Plume core region with high slip velocity is confined as a column of gas-phase which cannot be dispersed from the centerline. In higher slip velocity cases, TKE production by the buoyancy perturbation happened in the smaller region rather than smaller slip velocity case, but enhanced mean shear around inner plume core has a dominant contribution on the production of TKE.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A2B2007117, No. 2018H1A2A1063308).

4:53PM G28.00006 Numerical Simulation of Bubbly Flow and Underwater Acoustics under Breaking Waves based on a Coupled Resolved and Subgrid Scale Bubble Method, QIAN GAO, Department of Mechanical Engineering and Saint Anthony Falls Laboratory, University of Minnesota, GRANT DEANE, Marine Physical Laboratory, Scripps Institution of Oceanography, University of California, San Diego, LIAN SHEN, Department of Mechanical Engineering and Saint Anthony Falls Laboratory, University of Minnesota — Bubbles in breaking waves play an important role in many oceanography processes, including the bubble-mediated air-sea gas transfer, production of ambient wave noise, and marine aerosol generation. To study the bubble entrainment and breaking wave acoustics generation processes, we developed a coupled resolved and subgrid scale bubble simulation method. For bubbles greater than the grid size, which are considered resolved bubbles, we use a coupled level set and volume fraction functions. Bubbles smaller than the grid size, which are called subgrid scale bubbles, are treated with a four-way coupled polydisperse bubble model. Bubble-liquid interaction is accounted for by interfacial forces and void fraction. An underwater noise model is implemented for the generation of noise by wave breaking. It was found that our numerical method can capture the bubble size spectrum and wave noise accurately compared with experimental observations.

5:06PM G28.00007 Spray Atomization Using Bubbles Generated by a Two-Phase Counterflow Mixing Layer, ALISON HOXIE, ERIC JOHNSON, VINOD SRINIVASAN, PETER ROHRBACH, SUO YANG, KRISHNA BAVANDLA, HONGLUAN ZHANG, University of Minnesota — In this study, we employ the well-established phenomenon of the onset of global modes and concomitant rapid breakdown of certain shear layer configurations to design an efficient two-fluid mixer. Low-density jets and countercurrent mixing layers exhibit strong global modes and elevated turbulence levels, leading to rapid mixing, which is relatively insensitive to the viscosity-mediated mean shear stresses. By arranging a liquid and an atomizing gas (air) to satisfy the requirements for the onset of global modes, we are able to demonstrate efficient atomization. It is hypothesized that the observed droplet diameter is proportional to the diameter of bubbles that are created in a two-phase mixing layer inside the atomizer, which depends on the wavelength of the unstable global mode. The droplet data are shown to be relatively insensitive to viscosity. To partially test the hypothesis, Direct Numerical Simulations (DNS) are carried out using Eulerian-Eulerian Volume of Fluid (VOF) approach. Gas and liquid are considered as compressible fluids with perfect gas and perfect fluid equations of state, respectively. Schiller Naumann interphase drag model is used to capture the dynamics of gas bubbles in liquid, and surface tension is considered.
patterns. Such characterization has the potential to improve dynamic anti-fouling mechanisms. Through field experiments and laboratory analysis, we examine the interactions between fluid flow structures, interfaces, and biofouling growth. Various researchers have investigated select hydrodynamic effects, a comprehensive picture linking flow features to fouling development is elusive. As a consequence of both biological and environmental conditions, with the latter including a variety of possible fluid mechanic phenomena. While fuel costs, and even small amounts of fouling can interfere with scientific instruments in the field. It is known that biofouling growth is a consequence of both biological and environmental conditions, with the latter including a variety of possible fluid mechanic phenomena. While various researchers have investigated select hydrodynamic effects, a comprehensive picture linking flow features to fouling development is elusive. Through field experiments and laboratory analysis, we examine the interactions between fluid flow structures, interfaces, and biofouling growth patterns. Such characterization has the potential to improve dynamic anti-fouling mechanisms.

5:19PM G28.00008 Effect of Probe Inserts on the Local Void Fraction in a Bubble Column1, ARUSHI TIWARI, THOMAS J. BURTNETT, THEODORE J. HEINDEL, Iowa State University, EXPERIMENTAL MULTIPHASE FLOW LAB TEAM — Bubble columns are found in many process industries where a gas is bubbled through a liquid to promote mixing, separation, and/or reactions. The local void fraction (also referred to as a gas holdup or volumetric gas fraction) is an important parameter used to quantify flow conditions, and is one measure used to validate computational fluid dynamic simulations. The local void fraction is commonly measured with an inserted probe, but the presence of a probe can also modify local flow conditions and local void fraction. This study uses X-ray computed tomography to quantify the effect different probe geometries have on the time-average local void fraction. The results indicate the local void fraction is slightly higher when probes are inserted into the bubble column, but the overall void fraction is unaffected.

1This work was sponsored by the Office of Naval Research under grant number N00014-18-1-2319.

Sunday, November 24, 2019 3:48PM - 5:32PM – Session G29 Biological Fluid Dynamics : Animals 611 - Jamey Jacob, Oklahoma State University

3:48PM G29.00001 Examining the effects of hydrodynamic features on biofouling growth and suppression, LENA DUBITSKY, MARK MENESSES, Boston University, JESSE BELDEN, Naval Undersea Warfare Center - Division Newport, JAMES BIRD, Boston University — The growth of biofouling organisms such as algae, barnacles, and mussels on submerged surfaces is a ubiquitous phenomenon and generally undesirable. Heavy biofouling on ships leads to increased drag and subsequent fuel costs, and even small amounts of fouling can interfere with scientific instruments in the field. It is known that biofouling growth is a consequence of both biological and environmental conditions, with the latter including a variety of possible fluid mechanic phenomena. While various researchers have investigated select hydrodynamic effects, a comprehensive picture linking flow features to fouling development is elusive. Through field experiments and laboratory analysis, we examine the interactions between fluid flow structures, interfaces, and biofouling growth patterns. Such characterization has the potential to improve dynamic anti-fouling mechanisms.

4:01PM G29.00002 Chemical Interactions around Pulsing Soft Corals1, MATEA SANTIAGO, University of California, Merced, LAURA MILLER, University of North Carolina at Chapel Hill, SHILPA KHATRI, University of California, Merced — A subset of sessile Octocorals (Family Xenidae) actively and almost constantly pulse their tentacles. Experimental results indicate that the pulsating facilitates the photosynthesis of the symbiotic algae that live on the Octocorals. It is hypothesized that a significant source of the corals energy is the byproduct of the photosynthesis by the symbiotes. We model the photosynthesis of the symbiotic algae as a gas exchange of carbon dioxide to oxygen, where the coral tentacles are modeled as a source and sink of chemical concentrations. Additionally, the fluid-structure interaction of the pulsing corals, modeled using the immersed boundary method, is coupled to these chemical concentrations. We will present numerical simulations with varying parameters which have been used to gain insight to the complex interactions between the pulse driven fluid flow and the surrounding chemical concentrations.

1NSF PHY grant #1505061 and NSF DMS grant #1853608

4:14PM G29.00003 Mixing and pumping functions in a zebrafish larval intestine1, KENJI KIKUCHI, Department of Finemecanics, Graduate School of Engineering, Tohoku University, HYEONGTAK NOH, KEIKO NUMAYAMA-TSURUTA, Graduate School of Biomedical Engineering, Tohoku University, TAKUJI ISHIKAWA, Department of Finemecanics, Graduate School of Engineering, Tohoku University — Transportation phenomena in the gut are extremely important for digestive, metabolism and absorption in nutrient uptake. The function of mixing and pumping in the intestine, which is a relatively larger vessel in the tracts of the body, has been partially understood in the physiological and medical fields, but not been fully clarified in the physical and mechanical aspects. The flow in the intestine has been recently focused on a distribution of gut flora concerning inflammably bowel disease, diabetes, and cancer. Even though quasi-static distribution analysis of the gut flora has been developed using a next-generation sequencer for medical diagnostics, but mechanical reasons for medical and surgical therapies have not been approached due to invisibility in the body. Here, we proposed in vivo real-time intestinal flow measurement in the larval zebrafish intestine, which has justified similar constriction anatomically and genetically, using a fluorescent particle tracking velocimetry for analysis of mixing and pumping functions of the posterior and interior intestines. Pclt number in the intestines led us to our mechanical understanding; the mixing and pumping functions were crossing over after meal in the zebrafish larva.

1JSPS KAKENHI (Grant No. 17H00853 and 19H02059)

4:27PM G29.00004 Pump Function of C. elegans Pharynx in Highly Viscous Environments, YUKI SUZUKI, KENJI KIKUCHI, Department of Finemecanics, Graduate School of Engineering, Tohoku University, KEIKO NUMAYAMA-TSURUTA, Graduate School of Biomedical Engineering, Tohoku University, TAKUJI ISHIKAWA, Department of Finemecanics, Graduate School of Engineering, Tohoku University — A nematode C. elegans is a filter feeder, which lives in various viscous habitats such as soil and rooting fruits. C. elegans draws a suspension of food bacteria and separates them from the solvent water by using the pharyngeal pump. Former studies have proposed the mechanism of the food condensation only in low viscosity environments. Although C. elegans lives mostly in highly viscous habitats, few studies have investigated the food condensation in highly viscous conditions. Hence, it is not clear how C. elegans can eat bacteria to survive in highly viscous environments. In this study, we investigated the effect of viscosity on the survival of worms and the pump function of the pharynx in highly viscous conditions. We found that the survival rate of worms diminished with increase in viscosity. We also found that the pump function weakened due to higher viscosity while the pump power rose with increase in viscosity. This result suggests that the amount of ingested food declined with increase in viscosity since the pharyngeal pump could expand and contract inadequately in high viscosity. Finally, our results indicate that decrease in the survival rate of worms would be related with decline in the amount of food ingested by the pharyngeal pump in high viscosity.

4:40PM G29.00005 Snail feeding at the air-water interface, DAIISUKE TAKAGI, SOYOUIN JOO, ROBERT COWIE, University of Hawaii at Manoa, SUNGYON LEE, University of Minnesota, SUNGHWAN JUNG, Cornell University — Apple snails exhibit an intriguing feeding behavior at the air-water interface: they deform the foot to set up a funnel-like structure with surface waves traveling radially inwards. We report quantitative measurements of the resultant flow generated on and around the snails. Our observations reveal that distant food particles floating on the interface are effectively drawn in and collected at the center of the funnel. We develop a mathematical model based on lubrication theory to explore plausible physical mechanisms driving the entire system. The snails efficient feeding strategy offers a great source of inspiration for engineering devices designed to drive and control particles along any interface.
Additionally, a cross-wind is implemented to model the effect of the head-wind generated by the forward swimming of the dolphin. A two-phase flow model resolves the entrainment of the mucus in the expelled jet. A combined program of in-situ measurements, experimental setups, and computational simulations has been designed. This study comprises the analysis of their mucus contained in their breath. To capture a dolphin’s breath in the wild, an Unmanned Aerial Systems (UAS) can fly “through” the breath when expelled thus the extent of the exhaled breath is required to properly design the UAS platform. This jet is impulsive, unsteady, two-phase, and in cross flow. A mechanical device has been designed and fabricated to simulate this type of jet for use in wind tunnels and ultimately for UAS aircraft trials. Requirements for this simulator were obtained using three separate dolphins under human care of varying age, weight, and gender by taking high-speed videos of the dolphin’s breath in two planes. PIV measurements were calculated from the videos and used to guide the development of the specialized jet simulator. In addition, existing mass flow data from measurements of dolphins show that these breaths vary from 20-140 liters per second in a time duration of 0.26-0.31 seconds. These requirements were used to design the biologically inspired two-phase jet. Flow measurements of the blow-hole jet dynamics are compared with in-situ field data of actual dolphins.

Dolphin Quest

5:06PM G29.00007 Development of a Simulator to Mimic a Dolphin Blowhole Jet Flow Field

CHRIS BARTON, RICHARD GAETA, ALVIN NGO, MITCHELL FORD, ARVIND SANTHANAKRISHNAN, AARON ALEXANDER, JAMEY JACOB, Oklahoma State University — The health of bottlenose dolphins can be monitored by marine biologists through an analysis of their mucus contained in their breath. To capture a dolphin’s breath in the wild, an Unmanned Aerial Systems (UAS) can fly “through” the breath when expelled thus the extent of the exhaled breath is required to properly design the UAS platform. This jet is impulsive, unsteady, two-phase, and in cross flow. A mechanical device has been designed and fabricated to simulate this type of jet for use in wind tunnels and ultimately for UAS aircraft trials. Requirements for this simulator were obtained using three separate dolphins under human care of varying age, weight, and gender by taking high-speed videos of the dolphin’s breath in two planes. PIV measurements were calculated from the videos and used to guide the development of the specialized jet simulator. In addition, existing mass flow data from measurements of dolphins show that these breaths vary from 20-140 liters per second in a time duration of 0.26-0.31 seconds. These requirements were used to design the biologically inspired two-phase jet. Flow measurements of the blow-hole jet dynamics are compared with in-situ field data of actual dolphins.

Dolphin Quest

5:19PM G29.00008 Two-Phase Computational Fluid Dynamics Simulations of Dolphin Blowhole Expulsion Jets

AARON ALEXANDER, RICHARD GAETA, NGO ALVIN, MITCHELL FORD, JASON BRUCK, HALEY OBRIEN, Oklahoma State University — Monitoring the well-being of the wild dolphin population poses a challenge for biologists. While dolphins in human care can be trained to provide biological samples for monitoring, other methods must be utilized to obtain samples from wild dolphins. It is known that the mucus found in the flow generated from dolphins blowholes can be tested for hormones that help understand the current health status of the dolphin. Yet, the emitted jets from dolphin blowholes have not been well characterized. In order to understand these jets so that adequate samples may be obtained by Unmanned Aerial Systems (UAS) without spooking the dolphins, a combined program of in-situ measurements, experimental setups, and computational simulations has been designed. This study comprises the computational simulation leg of the effort and uses high fidelity scans of a dolphin respiratory system to create a computational fluid dynamic (CFD) replication of the jet emitted from the blowhole. A two-phase flow model resolves the entrainment of the mucus in the expelled jet. Additionally, a cross-wind is implemented to model the effect of the head-wind generated by the forward swimming of the dolphin.


3:48PM G30.00001 The Dynamics of Vesicles Driven through Closed Constrictions by Molecular Motors

YOUNGMIN PARK, THOMAS FAI, Brandeis University — Dendritic spines are postsynaptic processes that often appear in excitatory synaptic connections of principle neurons throughout the brain. Normal function depends on regular replenishment of various membrane proteins, which are transported on vesicles that squeeze through the spine constriction. In the present study, we explore the movements of vesicles in a reduced model of vesicle transport. The model predicts vesicle movement given motor transport parameters such as the ratio of competing motor forces and constriction geometry. We find that for moderately equal motor ratios and moderate constrictions, the movement may be bidirectional, and velocities gain or lose stability through sets of saddle-node bifurcations. For sufficiently tight constrictions, stable velocities vanish through a cusp bifurcation, resulting in one globally stable velocity. We explore the effects of noise on this system, and establish preliminary conditions for general force-velocity curves for vesicle trafficking.

4:01PM G30.00002 Cerebrospinal fluid influx is the earliest contributor to brain edema following stroke

JEFFREY TITHOF, University of Rochester, HUMBERTO MESTRE, TING DU, AMANDA SWEENEY, GUOJUN LIU, University of Rochester Medical Center, LOGAN BASHFORD, EDNA TORO, DOUGLAS KELLEY, University of Rochester, MAIKEN NEDERGAARD, University of Rochester Medical Center — Stroke is one of the leading causes of death worldwide, affecting 10 million people annually. The most detrimental complication is cerebral edema, which is the abnormal accumulation of fluid, thought to enter the brain from the blood. However, we demonstrate that rapid influx of cerebrospinal fluid (CSF) is the principal mechanism for brain edema in the first several minutes following stroke. Specifically, we block a large artery in the brain of a living mouse and then image fluorescent CSF tracers (infused prior to the stroke). By performing particle tracking velocimetry and front tracking velocimetry, we quantify CSF influx at both microscopic and brain-wide scales, respectively. Rapid influx of CSF occurs along the glymphatic pathway, which includes perivascular spaces (annular channels surrounding arteries). Our measurements demonstrate that constriction of arteries following stroke increases the effective size of the perivascular spaces and drives influx of CSF. Our results may lead to novel treatment strategies for stroke and suggest that glymphatic edema may be an important contributor to other acute brain pathologies, such as traumatic brain injury.

1This work is supported under NIH grant 1RF1AG057575-01
4:14PM G30.00003 Hydraulic resistance of periarterial spaces in the brain

1. JOHN H. THOMAS, JEFFREY TITHOF, DOUGLAS H. KELLEY, HUMBERTO MESTRE, MAIKEN NEDERGAARD, University of Rochester — Periarterial spaces (PASs) are annular channels around arteries that carry a flow of cerebrospinal fluid (CSF) into the brain, bringing in nutrients and sweeping away metabolic waste. In vivo observations reveal that PASs are not concentric circular annuli, as often assumed, but instead are oblate and eccentric. We model the PAS cross-section as a circle (artery) surrounded by an ellipse (outer wall), and vary the area, oblateness of the ellipse, and eccentricity of the circle relative to the ellipse. This model can match observed shapes of PASs quite well. For each shape, we determine the velocity profile for steady, laminar flow and compute the corresponding hydraulic resistance. The minimum hydraulic resistance (maximum flow rate) for a given cross-sectional area occurs when the ellipse is elongated and intersects the circle, dividing the PAS into two lobes, as is common around pial arteries. If both boundaries are circular, the minimum hydraulic resistance occurs when the eccentricity is large, as is common around penetrating arteries. We show that the actual shapes of PASs are nearly optimal, offering the least hydraulic resistance for their size: this may well represent an evolutionary adaptation that maximizes the clearance of metabolic waste from the brain.

1This research was supported by NIH/National Institute of Aging grant RF1 AG057575-01 and by Army Research Office grant MURI W911NF1910280

4:27PM G30.00004 Peristaltic Pumping in an Elliptical-Annulus Model of a Perivascular Space

1. J. BRENNEN CARR, JOHN H. THOMAS, JESSICA K. SHANG, Department of Mechanical Engineering, University of Rochester — Cerebrospinal fluid enters the brain parenchyma through a network of perivascular spaces (PVSs). The flow removes metabolic waste and its disruption is associated with stroke and neurodegenerative diseases. Recent in vivo experiments show that the flow is peristaltically pumped by traveling waves along the arterial wall. We simulate this perivascular pumping in three-dimensional models of the PVS using a finite element solver. The PVS is idealized as an annulus with a circular inner wall and an elliptical outer wall; a sine-wave displacement is propagated along the inner wall. We examine the effects of shape and eccentricity of the annulus and the amplitude and wavelength of the wave on velocity profiles and net flow rates. In contrast to the concentric circular annulus, in which the flow is axisymmetric, the velocity fields in an elliptical or eccentric annulus have an azimuthal velocity component. The net flow is in the direction of the propagated wave and is greater in elliptical and eccentric annuli than in a concentric circular annulus with the same cross-sectional area.

1This work is supported under NIH grant 1RF1AG057575-01

4:40PM G30.00005 Strain accumulation visco-elastic ventriculomegaly hypothesis for the onset of idiopathic Normal Pressure Hydrocephalus (iNPH)

1. STEPHANIE SINCOMB, University of California, San Diego, VICTOR HAUGHTON, School of Medicine- University of Wisconsin, Madison, ANTONIO SANCHEZ, ERNESTO CRIADO-HIDALGO, JUAN C LASHHERAS, University of California, San Diego — Idiopathic Normal Pressure Hydrocephalus (iNPH) also known as Chronic Adult Hydrocephalus is a syndrome characterized by ventriculomegaly, an enlargement of the brain ventricles containing cerebrospinal fluid (CSF), in the absence of elevated intracranial pressure (ICP). Symptoms of iNPH include urinary incontinence, disturbed gait, and dementia. It is most prevalent in the elderly population while extremely underdiagnosed. This condition has been subject of considerable studies, but the question remains of why while the ICP remains normal the ventricles continue to dilate despite free communication between the ventricles and the subarachnoid space. Understanding the mechanisms leading to iNPH is fundamental for its early detection and treatment. This is clinically significant as iNPH is the only potentially reversible neurodegenerative disease. Through Magnetic Resonance Elastography (MRE) measurements of the viscoelastic properties of the brain parenchyma and analytical modeling, we investigate the hypothesis that the reduction in the stiffness of the periventricular white matter and/or a decrease in its permeability leads to a gradual accumulation of the CSF in the brain ventricles and its subsequent enlargement.

4:53PM G30.00006 Numerical simulations and experiments of CSF flow in the spinal canal

1. CANDIDO GUTIERREZ-MONTES, Universidad de Jaen, WILFRIED COENEN, JENNA J LAWRENCE, Univ of California - San Diego, CARLOS MARTINEZ-BAZAN, Universidad de Jaen, ANTONIO L SANCHEZ, JUAN C LASHHERAS, Univ of California - San Diego — Besides the oscillatory velocity driven by the cardiac and respiratory cycles, CSF flow in the spinal canal exhibits a slow steady Lagrangian motion comprising steady-streaming and Stokes-drift contributions, described in recent analytical work. Associated subject-specific descriptions of this bulk flow have revealed the existence of closed Lagrangian recirculating vortices in the lumbar, thoracic, and cervical regions. The structure of these vortices and their relevance in connection with intrathecal drug delivery (ITDD) applications are further investigated in the present study by transient numerical simulations employing a dynamic-mesh fluid-structure interaction method to account for the deformation of the dura membrane. The results show excellent agreement with the previous analytical predictions, which are further corroborated by accompanying in-vitro experiments. The description is extended to account for the buoyancy-driven motion emerging in ITDD procedures as a result of the density mismatch between the drug and the CSF.

1Supported by National Science Foundation through Award Number 1853954

5:06PM G30.00007 Modeling Filarial Worm Migration in Lymphatic System

1. KI WOLF, J. BRANDON DIXON, ALEXANDER ALEXEEV, Georgia Institute of Technology — Lymphatic filariasis is a prevalent condition in tropical countries and is caused by intrusion of parasitic worms such as W. bancrofti, which co-parasite between humans and mosquitoes. These parasites can leave debilitating lifelong damage to the lymphatic system. The biophysical traits that enable filarial survival within their lymphatic niche is unknown. To address this problem, we develop a fully coupled, three-dimensional fluid-solid computational model of parasite movement inside the lymphatic vessel. The model is employed to probe parasite movement under various lymph flow conditions. Worm and lymphatic valve parameters are varied to examine the parasite interactions with lymphatic valves. We compare our simulations with recent experiments on filarial movement inside the lymphatic system (Kilaraki 2019). Our results provide insights into the mechanisms behind intralymphatic migration of parasitic worms during the onset and persistence of infection in filariasis.

1Financial support from the National Science Foundation (CMMI-1635133) is gratefully acknowledged.
Motion. We calculate the velocity-velocity correlation functions and the effective diffusivity of passive tracers, and reveal their distinction is the appearance of collective motion in self-propelled particles suspended in a fluid observed in recent experiments and simulations: Milling motions with a hollow core are also observed.

\[ \chi \] of the microswimmers, and the fraction of active microswimmers present in the system (\( \chi \)). We then explore the specificities of the system skull-CSF-brain (flow of the CSF in and out of the skull through the spinal cord, confinement of the CSF...). From there, we show that the energy stored in one bubble only depends on (\( \alpha, \tau \)) and build a phase diagram of the damaging capacity of the bubbles on the brain as a function of (\( \alpha, \tau \)), in good agreement with the prediction of the WSTC.

Sunday, November 24, 2019 3:48PM - 5:32PM – Session G31 Biological Fluid Dynamics: Micro-Swimmer General II

3:48PM G31.00001 Clogging of microswimmers at a constriction.¹, PHILIPPE PEYLA, MARVIN BRUN-COSME-BRUNY, University Grenoble Alpes, SALIMA RAFAI, CNRS, ANDRE FOERTSCH, WALTER ZIMMERMANN, Bayreuth University, THEORETICAL PHYSICS TEAM, LIPHY TEAM — We study the clogging of a suspension of photosensitive microswimmers [Chlamydomonas Reinhardtii (CR)] moving to a constriction in a microfluidic device. Swimming cells are fleeing light and accumulate at a gate that is twice larger than a CR. We study the statistics of times between two successive egresses. Our results fall in the general framework of clogging obtained for panicking pedestrians at a gate or granular materials at the exit of a silo: the survival function obeys a power law decrease with times. Our results also show a faster-is-slower effect: when cells are faster, clogging is increased. This phenomenon - very well known in crowd evacuation during a panic - unveils the role of tangential friction between cells at the constriction where unusual densities are reached. Our experimental observations are supported by lattice Boltzmann simulations.

¹German French University

4:01PM G31.00002 Bacterial magneto-convection¹, ALBANE THERY, DAMTP, Cambridge University, LUCAS LE NAGARD, JEAN-CHRISTOPHE ONO-DIT-BIOT, CECILE FRADIN, KARI DALNOKI-VERESS, McMaster University, ERIC LAUGA, DAMTP, Cambridge University — Magnetotactic bacteria are prokaryotic microswimmers that synthetize magnetosomes, which are chains of nano-magnets in their cytoplasm. Under external magnetic fields these cells are subject to magnetic torques and thus passively align along magnetic lines, providing a simple control mechanism. We use microfluidics experiments to show that collective motion arises from an initial uniform distribution of magnetotactic bacteria under confinement when an external magnetic field is applied. Dense suspensions of magnetotactic bacteria of strain AMB1 are driven toward a glass capillary wall by a magnetic field perpendicular to the channel walls. Their initial random spacing on the surface becomes unstable due to attractive hydrodynamic interactions between swimmers. These interactions result in bacterial magneto-convection: bacterial plumes perpendicular to the wall emerge spontaneously and develop into self-sustained convection cells. The plumes grow and merge and their dynamics is studied experimentally by measuring their wavelength and the flow they generate. Using a numerical model based on hydrodynamic singularities, we are able to capture quantitatively the instability observed in the cell suspension and reproduce the flow field as well as the long-time clustering dynamics.

¹Funded by ERC (No. 682754 to EL) and the Natural Science and Engineering Research Council of Canada

4:14PM G31.00003 Phase transitions in the system of active and passive microswimmers, NAVEEN KUMAR AGRAWAL, PALLAB SINHA MAHAPATRA, Indian Institute of Technology Madras, Chennai (600036), India — We study the steady-state phases and their transition in a system of active and passive microswimmers. The active and passive microswimmers are initially, randomly distributed in a fluidic medium inside a square enclosure. The active microswimmers move by a constant magnitude self-propulsion force. Whereas, the passive microswimmers have no self-propulsion force, and they move by force exerted by the fluid and the other neighboring microswimmers. A microswimmer's interactions with other microswimmers and the fluidic medium govern the direction of the exerted thrust on it. We have used a discrete particle model to solve the governing equations. Here, the hydrodynamic interaction is modeled as Stokes drag. The phase transition depends on the coordination coefficients (identified by a parameter \( \chi \)) of the microswimmers, initial states of the microswimmers, and the fraction of active microswimmers present in the system (\( \rho \)). At low \( \chi \), the microswimmers exhibit a random motion. For the higher \( \chi \) values, the phase transits from random motion to the milling phase, where microswimmers rotate around the core. Milling motions with a hollow core are also observed.

4:27PM G31.00004 Correlations in microswimmer suspensions, VIKTOR SKULTETY, ALEXANDER MOROZOY, University of Edinburgh, UK — Recent years witnessed a significant interest in physical, biological and engineering properties of self-propelled particles, such as bacteria or synthetic microswimmers. The main distinction of this ‘active matter’ from its passive counterpart is the ability to extract energy from the environment and convert it into directed motion. One of the most striking consequences of this distinction is the appearance of collective motion in self-propelled particles suspended in a fluid observed in recent experiments and simulations: at low densities particles move around in an uncorrelated fashion, while at higher densities they organise into jets and vortices comprising many individual swimmers. Here, we present a novel kinetic theory that predicts the existence of strong correlations even below the transition to collective motion. We calculate the velocity-velocity correlation functions and the effective diffusivity of passive tracers, and reveal their non-trivial density dependence. The theory is in quantitative agreement with our recent Lattice-Boltzmann simulations (J. Stenhammar et al., Phys. Rev. Lett. 119, 028005 (2017)) and captures the asymmetry between pusher and puller swimmers below the transition to collective motion.
4:30PM G31.00005 Auto-phoretic nanorods driven up the wall by gravity, QUENTIN BROSSEAU, New York University - Courant Institute, FLORENCIO BALBOA USABIAGA, Flatiron Institute, ENKELEIDA LUSHI, New Jersey Institute of Technology, YANG WU, New York University, LEIF RISTROPH, New York University - Courant Institute, MIKE WARD, New York University, MIKE SHELLEY, JUN ZHANG, New York University - Courant Institute — Gravitaxis is the directed upward motion of microorganisms against gravity, and is observed for a few ciliated organisms like Chlamydomonas, Euglenas or Paramecium. Lacking a dedicated sensor, their gravitactic response relies on bottom-heavyness or shape anisotropy to induce a bias in their swimming direction. Here we study the gravitaxis of heavy self-electrophoretic Janus nanorods that move upwards on a steeply inclined substrate. Comparisons in experiments and simulations between homogeneous and bottom-heavy nanorods reveal two mechanisms contributing to the gravitactic response of the latter: a buoyancy torque and hydrodynamic interactions with the wall. We show that lubrication forces induce an effective fore-aft asymmetry on nanorods that reinforces the orientation bias to move up the steep wall against gravity.

4:50PM G31.00006 Transport and Dynamics of Swimming Microorganisms in Time-Periodic Flows¹, RANJIANGSHANG RAN, BOYANG QIN, BRENDAN BLACKWELL, PAULO ARRATIA, University of Pennsylvania — Microorganisms often need to navigate through complex fluid environments to successfully feed and reproduce. Examples include algae in oceans and lakes, bacteria in gastrointestinal tracts, and during the production of food and vaccines. Here, we experimentally investigate the transport and mixing of swimming E. coli in two-dimensional time-periodic flows using particle tracking velocimetry and dye mixing experiments. Mixing is assessed by computing stretching fields and finite-time Lyapunov exponents (FTLE) from experimentally measured velocity fields. Velocimetry data shows that bacteria lowers the peak velocity and vorticity of the flow. This result is a function of bacteria concentration and flow geometry; that is, flow separatrices hinder mixing compared to passive particles likely due to bacteria trapping. Stretching fields show that bacteria shorten the average wavelength of Lagrangian coherent structures (LCSs). Overall, mixing can increase or decrease by the addition of active particles (i.e. swimming bacteria) but there seems to be a non-trivial dependence on Reynolds number, path length, flow geometry, and bacteria volume fraction.

¹This work is supported by NSF-DMR-1709763

5:06PM G31.00007 Route to bacterial swarming¹, XIANG CHENG, YI PENG, ZHENGYANG LIU, University of Minnesota — Collective motions of active fluids such as bird flocks, fish schools and bacterial swarms demonstrate the intriguing emergent behaviors of nonequilibrium systems. While moving at low density, active entities in an active fluids move collectively with its neighbors at high density, exhibiting strong orientational order at a scale of order larger than the size of individual entities. Although such a disorder-order nonequilibrium phase transition has been previously studied, the detailed kinetics of this transition has not been systematically explored in experiments. Here, using light-controlled E. coli, whose locomotion can be reversibly controlled by light, we experimentally study the kinetic pathway of the swarming transition in 3D bacterial suspensions. The phase diagram of bacterial swarming as a function of bacterial concentration, the velocity of active swimmers and the number fraction of active swimmers is systematically mapped. Moreover, we identify different kinetic pathways for the swarming transition depending on the control parameters. Our results reveal the route to the emergent bacterial swarming and provide new insights into the nonequilibrium phase transition in active fluids.

¹This work is funded through ERC project no 716712

5:19PM G31.00008 Hydrodynamic interactions of Chlamydomonas with a solid surface¹, ABEL-JOHN BUCHNER, KOEN MULLER, DANIEL TAM, Technical University of Delft — Motile unicellular organisms swim through complex environments and often interact with solid surfaces. Their swimming is influenced by the proximity to solid substrates, through hydrodynamic and steric interactions. These interactions directly influence the cell population density distribution and the residence time in the vicinity of the surface. It is predicted that cell-cell interactions remain unclear and previous experimental studies have often been limited to two-dimensional flow cells, which confine the trajectories of the swimming cells. Here, we investigate the interaction of free-swimming cells with surfaces in an otherwise unconstrained three-dimensional flow chamber. Our tracking experiments focus on the model “puller” organism Chlamydomonas reinhardtii. Swimming cells are recorded simultaneously by four separate cameras and triangulated in three-dimensions. Kinematic statistics are calculated from approximately 30,000 swimming tracks. Our results provide evidence of the existence of a long-range hydrodynamic interaction, which induces orbiting behaviour in the near-surface region.

¹This work is funded through ERC project no 716712

Sunday, November 24, 2019 3:48PM - 5:32PM — Session G32 Biological Fluid Dynamics: Plant Biomechanics

3:48PM G32.00001 Leaf-to-leaf spore dispersal of rust induced by rainsplash¹, HYUNG-GON PARK, Virginia Tech, SEUNGHO KIM, Cornell University, HOPE GRUSZEWSKI, DAVID SCHMALE III, Virginia Tech, SUNGHWAN JUNG, Cornell University, JONATHAN BOREYKO, Virginia Tech — Rainsplash is an important dispersal mechanism for plant pathogens, but the underlying fluid dynamics are poorly understood. We studied how spore-laden satellite droplets, ejected from rainsplash on a diseased leaf, can subsequently stick to healthy leaves. This natural process was mimicked by rebounding microscopic droplets from an angled superhydrophobic substrate onto an adjacent wheat leaf that was horizontally oriented. The droplets either skipped along the wheat leaf or became stuck, depending upon the droplet’s inertia, the orientation of the anisotropic leaf structure with respect to the droplet’s path of motion, and whether the leaf was untreated or sprayed with a fungicide. A model successfully demarcated skipping versus sticking behavior by comparing the droplet’s inertia against the surface energy required for dewetting to occur upon impact.

¹This work was supported by the USDA National Institute of Food and Agriculture (Award No. 2018-67013-28063) and by Virginia Tech’s BIOTRANS program.

3:48PM G32.00002 Leaf-to-leaf spore dispersal of rust induced by rainsplash
Comparisons with published laboratory experiments suggest that the inclusion of TD improves the prediction of sucrose transport speed. This mechanism of spore dispersal enables Sphagnum to carry its spores beyond the turbulent boundary layer where it grows so that they can be carried indefinitely by wind currents, which would not be possible if the spores were launched ballistically. Here we present a finite element analysis of the explosive spore discharge from Sphagnum capsules using ANSYS Fluent. We model the spore capsule as a pressurized cylinder with a disk shaped cap that is pushed by the fluid surrounding it which is taken to be incompressible. The capsule properties of the fluid during the vortex ring is created and models this case qualitatively different than the slug flow from a piston-driven system without a cap. By matching the trajectories of vortex rings in our models to high speed videos of capsule explosions we can determine the initial pressure of the capsule. Moreover by analyzing the flow of vorticity out of the capsule we can determine how the cap affects the optimality of vortex ring impulse and compare Sphagnum vortex rings to the optimal vortex rings produced by animals.

Wind-Induced Damage Propagation in a Branched Tree Forest.

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIP) (No. 2017R1A2B3005415).

Hydraulic regulation of air spreading in vascular plants using wetted cellulose membranes1, JOOYOUNG PARK, TAEOK GO, JONGEUN RYU, SANG JOON LEE, Pohang University of Science and Technology, BIOFLUID AND BIOMIMIC RESEARCH CENTER TEAM — Plants transport water through xylem vessels using strong negative pressure. Thus, the transport system becomes vulnerable to cavitation. For stable water transport, xylem structures have been evolved. Xylem vessels are interconnected by pit membranes consisting of cellulose fibers that function as hydraulic valves to regulate two-phase flows. In this study, the hydraulic roles of pit membranes to prevent spreading of air throughout xylem vessels were investigated using a model system consisting of channels embedding cellulose membranes. We hypothesized that pit membranes normally remain to be wetted in xylem vessels. We noticed in particular the hydraulic role of water film on air bubble spreading that has been overlooked previously. We elucidated the correlation of dynamic characteristics of air flow and pressure variations with the membrane thickness. As a result, the air spreading exhibits two types of dynamics: continuous and discrete spreading. In addition, the thickness of pit membranes affects the behaviors of water film captured by cellulose fibers. Thus, it is a crucial criterion for the reversible gating of further spreading of cavitation throughout xylem networks of under drought condition.

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIP) (No. 2017R1A2B3005415).

Superhydrophilic-superhydrophobic water harvester inspired by the wetting property of cactus stem1, NAMI HA, HYEJEONG KIM, SANG JOON LEE, Pohang University of Science and Technology, BIOFLUID AND BIOMIMIC RESEARCH CENTER TEAM — Cacti can survive in extremely dry environment thanks to their unique water collecting ability. In addition to the spine which is one of the water collection systems, cactus stem actually plays an important role for harvesting and storing the absorbed water. In this study, we investigate the flow interaction with a forest of fractal branched trees to predict the propagation of damage that occurs and investigate the resulting flow field in different storm conditions. Also, a tree breakage model is incorporated into our simulations to study the progressive damage propagation in each tree and in the forest.

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIP) (No. 2017R1A2B3005415).

Controlling intercellular flow through mechanosensitive plasmodesmata nanoneopores2, KAARE H. JENSEN, JAN KNOBLAUCH, Department of Physics, Technical University of Denmark, KARL OPARKA, Institute of Molecular Plant Sciences, University of Edinburgh, KEUNHWAN PARK, Department of Physics, Technical University of Denmark — In plants, plasmodesmata (PD) are nanopores that serve as channels for molecular cell-to-cell transport. Precise control of PD permeability is essential to regulate nutrient transport; moreover, it is involved in growth, tissue patterning, and defense against pathogens. Callose deposition modulates PD transport but little is known of the rapid events that lead to PD closure in response to tissue damage or osmotic shock. We propose a new mechanism of PD closure as a result of mechanosensing. Pressure forces cause the cell wall, or the dumbbell-shaped ER-desmotubule-complex, to be displaced from the equilibrium position, thus closing the PD aperture. Cell wall elasticity and the filamentous protein tethers that link the plasma membrane to the ER complex play a key role in determining the selectivity of the PD pore. This model of PD control compares favorably with experimental data on the pressure-generated closure of PD.

This work was supported by the National Science Foundation (NSF- AGS-1644382; NSF-IOS-175489).

Taylor Dispersion in Osmotically Driven Laminar Flows1, MAZEN NAKAD, Student, JEAN-CHRISTOPHE DROME, THOMAS P. WITELSKI, GABRIEL KATUL, Professor — Sucrose is among the main products of photosynthesis that are deemed necessary for plant growth and survival. It is produced in the mesophyll cells and translocated under positive pressure to different parts of the plant through a hydraulic network (phloem). Progress in understanding this transport mechanism remains fraught with experimental difficulties thereby prompting interest in theoretical approaches and laboratory studies. The Munch’s pressure flow model is considered one of the most commonly accepted hypotheses for describing the physics of sucrose transport in such systems. It is based on osmosis to build an energy potential difference between the source (leaf) and the sink (root). The flow responding to this energy potential is assumed laminar and described by the Hagen-Poiseulle equation. The work here will revisit such osmotically driven flow in tubes by including the effects of Taylor dispersion (TD) on mass transport. It is demonstrated that the time scale for sucrose transport is reduced when adding TD for low Munch number (defined by the ratio of axial to membrane resistance) whereas its effect will decrease for high Munch number. Comparisons with published laboratory experiments suggest that the inclusion of TD improves the prediction of sucrose transport speed.

This work was supported by the National Science Foundation (NSF- AGS-1644382; NSF-IOS-175489).
Oil-water separation using synthetic trees. VIVERITA UMASHANKAR, Virginia Tech, ARUN KOTA, Colorado State University, JONATHAN BOREYKO, Virginia Tech — In the world’s tallest trees, water evaporating from leaves generates enough suction to lift water over 100 m high. Transpiration can similarly be attained in synthetic trees by coupling nanoporous "leaves" with conduits mimicking xylem capillaries. Here, we demonstrate that by adding filters to the free ends of the xylem conduits, the hydraulic load generated by transpiration can be used for oil-water separation. Our synthetic tree was comprised of a vertical array of glass tubes of millimetric diameter, whose upper ends were embedded within a nanoporous ceramic disk and whose lower ends were attached to pre-wet cellulose acetate membranes supported by stainless-steel meshes. After saturating the synthetic tree with degassed water, its synthetic roots (filters) were submerged in a reservoir containing a hexadecane-in-water emulsion. The separation efficiency of liquid entering into the tree was quantified by density and contact angle measurements of permeate extracted from the glass tubes.

This work was supported by the NSF CAREER Award (CBET-1653631) and by Virginia Tech’s BioBuild program.

3:48PM G33.00001 Global modes in Taylor–Couette–Poiseuille Flow with a permeable inner cylinder. DENIS MARTINAND, Aix Marseille Univ, CNRS, Centrale Marseille, M2P2, Marseille, France, NILS TILTON, Department of Mechanical Engineering, Colorado School of Mines, Golden CO 80401 — The addition to a Taylor–Couette cell of an axial Poiseuille flow and a radial flow associated with a weakly permeable inner cylinder results, at a given rotation rate of the inner cylinder, in adjacent regions of the flow that are simultaneously stable, convectively unstable, and absolutely unstable, making this system fit for obtaining global modes of centrifugal instability. Critical conditions of the instabilities are obtained using the analytical frameworks of linear and non-linear global modes. Besides, dedicated Direct Numerical Simulations implementing the Darcy’s condition on the permeable cylinder are performed to assess the validity of these analyses. Three different set-ups are considered. Fluid injection, in the first set-up, or extraction, in the second, occur along the full length of the inner cylinder. In the third, fluid flux through the inner cylinder evolves from extraction to injection as cross flow reversal occurs. Though correctly predicting the nature and location of the wave-makers governing the global instability, and their critical conditions, the analyses do not explain, however, that the instabilities observed in the numerical simulations take the form of axial stacks of wave-packets characterized by step-ups and step-downs of the temporal frequency.

This material is based upon research supported by the Thomas Jefferson Fund of the Embassy of France in the United States and the FACE Foundation.

4:01PM G33.00002 Randomized Resolvent Analysis of Turbulent Separated Flow over a NACA0012 Airfoil. JEAN HELDER MARQUES RIBEIRO, CHI-AN YEH, KUNIHIKO TAIRA, University of California, Los Angeles — Singularity decomposition of a large resolvent operator for high-Reynolds number fluid flow is computationally and memory intensive. For this reason, the applications of resolvent analysis has generally been limited to 1D or laminar flow problems. We consider randomized analysis for fast computation of the dominant resolvent modes by passing a tall and skinny random test matrix to sketch the large resolvent operator. The operator is then projected onto the low-dimensional subspace spanned by the sketch. By performing SVD on this reduced resolvent matrix, we achieve significant reduction in computational and memory requirements compared to traditional techniques. We apply this randomized approach for a bi-global resolvent analysis on a turbulent mean flow over a NACA0012 airfoil at chord-based Reynolds number of 23,000. While the full resolvent operator is projected using only 10 out of the 750,000 bases, the leading gain, forcing and response modes are accurately captured. We also provide discussions on incorporating physical insights into the randomized algorithm for further computational alleviation.

This work was supported by ONR (N00014-16-1-2443), AFOSR (FA9550-18-1-0040), and ARO (W911NF-17-1-0118). We also thank the DoD HPC Modernization Program.

4:14PM G33.00003 Global instability mode in a baffled Von Karman flow. NICHOLAS WORTH, JAMES DAWSON, Norwegian University of Science and Technology — A low-frequency spectral peak is identified in velocity spectra measured close to the stagnation point of a baffled turbulent Von Karman swirling flow. This observation is consistent across a range of studied experimental datasets (3DSC Scanning-PIV and 2DSC Stereo-PIV measurements) acquired in two different facilities. An attempt is made to recognise the underlying velocity structure through an application of conditional averaging and Proper Orthogonal Decomposition. The structure that emerges takes form of a spherical region located near the tank’s centre of accelerated, unidirectional flow perpendicular to the axial direction. This structure is then observed to precess around the tank’s axis with the frequency of the spectral peak. The dynamics of the recognised feature is then studied via POD-Galerkin projection of the Navier-Stokes equations. This ultimately allows us to track origins of the structure down to the linear term of the projected equations, which is characterised by a pair of unstable eigenvalues. The structure can be, therefore, classified as a global instability mode. A simple sensitivity analysis shows that the characteristic frequency can be controlled via the mean shear in the radial plane.

4:27PM G33.00004 Flow Visualization and Velocity Spectra in a Low Viscosity Round Jet. VINOD SRINIVASAN, IAN WRIGHT, University of Minnesota — Low-density jets are known to exhibit strong non-linear global modes, whose frequency is dependent on the density ration, boundary layer thickness with a weak dependence on viscosity. The separate roles of viscosity gradients has not been investigated experimentally for free shear flows such as round jets. The present research documents the unstable response of a circular jet issuing into an ambient fluid of higher viscosity. Viscosity ratios (ambient-to-jet) of 1 to 40 and jet Reynolds numbers of 500 to 2000 are studied in density-matched, miscible fluids. The mode of breakdown is visualized using fluorescent dye and hydrogen bubble techniques, while the wavelength of the dominant mode is measured through hot-film anemometry. The spatial and temporal growth of instabilities is reported as dependent on viscosity ratio, Reynolds number, and jet shear layer inlet condition. The breakdown process is marked by the emergence of a sharp peak in the frequency spectrum at a distance of half a jet diameter downstream of the exit plane. This peak persists for about 5 diameters downstream, gradually decreasing in magnitude until indistinguishable from the background. The frequency of the peak depends on the viscosity ratio, for any fixed jet Reynolds number.
4:40PM G33.00005 Global mode induced by a symmetry-breaking in a split cylindrical cavity. JESUS O. RODRIGUEZ-GARCIA, Universidad de Navarra, SOLEDAD LE CLAINCHE, Universidad Politécnica de Madrid, JAVIER BURGUETE, Universidad de Navarra — We study experimentally the flow inside a closed cylinder split in two halves at the equator. When these two parts rotate in exact corotation regime the internal flow is essentially in solid-body rotation at the angular velocity of both sides. When a slight difference between the rotation frequencies is established a secondary flow is created due to the differential rotation between halves and restricted to the boundary layer. The behavior of this boundary layer is compared with theoretical and numerical results finding the "sandwich" structure of a Stewartson boundary layer. Time-dependent structures are observed near the cylindrical wall. Their behavior for different values of the Reynolds and the Rossby numbers are presented. A global recirculation mode is also found due to a symmetry-breaking induced between sides that appears because of a slight misalignment of the experimental setup, whose characteristics are compatible with the behavior of a processing cylinder. A HODMD analysis is performed finding relevant frequencies inside the flow that allow us to reconstruct the global mode.

1 FIS2017-83401-P

4:53PM G33.00006 Localized eigenmodes in a moving frame of reference representing convective instability. KOEN GROOT, Texas A&M University, SEBASTIEN NIESSEN, University of Liege — When representing convective instability mechanisms with the streamwise BiGlobal stability approach, results suffer from a sensitivity to the streamwise domain truncation length and boundary conditions. The presently proposed methodology resolves this sensitivity by considering a moving frame of reference. In that frame, the spectrum features discrete eigenvalues whose corresponding eigenfunctions decay exponentially in both the up- and downstream directions. Therefore, the truncation boundaries can be placed far enough that both variations in the domain length and artificial boundary conditions have no impact. The discrete nature of the spectrum enables the use of (non-)local stability methods to perform an independent approximation of the BiGlobal eigenvalues via global mode theory. We demonstrate that retrieving set-up-independent solutions in the stationary frame of reference is likely impossible for the flow case considered.

1 FNRS-FIRA fellowship for Sebastien Niessen

5:06PM G33.00007 Global Analysis of the Convective Instabilities in Laminar Shock-Wave/Boundary-Layer Interactions. SEBASTIEN NIESSEN, University of Liege, KOEN GROOT, Texas A&M University, STEFAN HICKEL, Technische Universiteit Delft, VINCENT TERRAPON, University of Liege — The interaction between a laminar boundary layer and an oblique shock wave is investigated through BiGlobal stability analysis. Previous studies have revealed the presence of a stationary mode (J.-Ph. Boin et al., TCFD 20(3), 2006, and J.-Ch. Robinet, JFM 579, 2007) and a convective instability mechanism (F. Guio et al., JFM 789, 2016). In the latter analyses, non-localized eigenmodes are obtained that are artificially affected by the finite size of the chosen computational domain. In the present work, we obtain localized wave packets, that are independent of domain size and truncation boundary conditions, by applying the stability analysis in a moving frame of reference. The long-time behavior is subsequently determined by time integration, which results in the propagation of the localized wave packets in the flow. Finally, we highlight the mechanisms constituting the convective instabilities through a Reynolds-Orr energy budget analysis. To obtain the unstable basic state solution to the compressible Navier-Stokes equations, we have coupled Direct Numerical Simulations to the selective frequency damping method.

5:19PM G33.00008 Linear Stability of Flow through a Compressor Stage. MIGUEL FOSAS DE PANDO, Department of Mechanical Engineering and Industrial Design, University of Cadiz, PETER J. SCHMID, Department of Mathematics, Imperial College London — Fluid systems of industrial interest often consist of a periodic assembly of identical subunits. For instance, a compressor stage in a jet engine is built by arranging a number of airfoil blades in annular rows, one rotating and one stationary. A typical analysis of these systems attempts to describe the flow dynamics by isolating a single subunit, i.e. a blade passage, and imposing periodic boundary conditions. Although this approach leads to problems that are easier to solve, it cannot account for global large-scale synchronization effects across multiple subunits. In this contribution, we first present a computational framework for the analysis of modal and non-modal stability of the full system, i.e. considering the contribution of each subunit to the global dynamics. This technique relies on the underlying properties of operators that are reminiscent of twisted Toeplitz matrices, which are in turn coupled through a time-dependent sliding interface. This framework will be then used to investigate the dynamics of the response of flow through an idealized compressor stage, consisting of a rotor and a stator, to small amplitude forcing.

1 The authors gratefully acknowledge financial support from MINECO AEI/ERDF EU (Grant No. DPI2016-75777-R).

Sunday, November 24, 2019 3:48PM - 5:19PM — Session G34 Flow Instability: Kelvin-Helmholtz

3:48PM G34.00001 Velocity Measurements and Phase Tracking in a Shock-Driven Multiphase Instability. VASCO O. DUKE WALKER, JOHN MIDDLEBROOKS, ROY ALLEN, WILLIAM MAXON, ALMUHNA SAHIR, Univ of Missouri Columbia, SAMRA KARABEGOVIC, Rock Bridge H.S. Columbia, JACOB A. MCFARLAND, Univ of Missouri Columbia — An experimental investigation has been performed to study physical phenomena induced by the impulsive acceleration of a heterogeneously seeded multiphase flow-field within a shock tube system. In order to achieve this, a cylindrical interface comprising of nitrogen, seeded with micron-sized acetone droplets, was generated within the shock tube's test section and accelerated by a planar shock wave. The nitrogen gas itself was saturated with acetone vapor tracer and mixed into the interface to prevent premature droplet evaporation. The development of both the dispersed and carrier phases was captured through a series of Planar Laser Mie Scattering (PLMS) and Planar Laser Induced Fluorescence (PLIF) images, respectively. In addition, lag effects between the phases were visualized and quantified. The results were compared to simulations for validation. This experimental investigation has a multitude of applications in a variety of scientific and engineering systems, with a particular relevance to systems that involve high-speed or shock-induced multiphase combustion.
4:01PM G34.00002 Liquid film dryout in vertical two-phase annular flow in a rectangular channel, ROMAN MORSE, University of Wisconsin - Madison, TIAGO MOREIRA, University of Sao Paolo / University of Wisconsin - Madison, KRISTOFER DRESSLER, University of Wisconsin - Madison, GHERHARDT RIBATSKI, University of Sao Paolo. LOUISE MCCARROLL, EVAN HURLBURT, Naval Nuclear Laboratory, GREGORY NELLS, ARGANTHAL BERSON, University of Wisconsin - Madison, HEAT TRANSFER RESEARCH GROUP, USP COLLABORATION, MULTIPHASE FLOW VISUALIZATION AND ANALYSIS LABORATORY, UW-MADISON COLLABORATION — The entire liquid-film dryout process in a vertical two-phase annular flow is characterized experimentally, from the initiation of the instability and its development up to the point where the film is completely removed. This system is made of two immiscible liquids in a microfluidic channel with a hydraulic diameter of 18 mm and aspect ratio of 1/3. The walls of the test section are made of glass coated with Fluorine-doped Tin Oxide (FTO). Heat fluxes up to 30 kW/m² are generated at the inner surface of the window by passing an electrical current through the FTO coating. Instantaneous pressure and temperature in the test section, temperature on the outer wall of the test section, and high-speed videos were recorded simultaneously during the dryout event. In addition, the state (wet or dry) of the heated surface was measured using thermoreflectance as a function of time. During the inception of dryout, dry patches on the heated surface may re-wet intermittently. The ability of the surface to rewet near dryout is studied under steady state and pulsatile conditions.

4:14PM G34.00003 Backflow at liquid-gas interface driven by Marangoni effects, HONGYUAN LI, PENCYOU LV, HUILING DUAN, Peking University — Liquid-gas interface on submerged structured surfaces is ubiquitous in nature and has various practical applications. These interfaces, however, are subject to non-uniform surface tension induced by surfactant concentration gradient. Careful PIV experiments with surfactant-laden fluorescent particle solutions show that even trace surfactants have significantly decreased on slip velocity than numerical predictions for surfactant-free flow, and even can produce negative slip velocity (i.e., backflow). We present the mechanism of backflow, which is induced by surfactant concentration gradient. Meanwhile, we investigate the characteristics of backflow. This work is expected to be essential for the design of slippery surfaces and microfluidic devices that are unaffected by Marangoni stress, and it can provide guidance for correcting PIV measurements, especially for the case of liquid-gas interface.

4:27PM G34.00004 Interfacial structure of upward gas-liquid annular flow in an inclined pipe, ADAM FERSHTMAN¹, Tel-Aviv University, LULKAS ROBERS, HORST-MICHAEL PRASSER, ETH Zurich, DVORA BARNEA, LEV SHEMEN, Tel-Aviv University. — In this study, the instantaneous circumferential distribution of the liquid film thickness is examined in upward annular flow of various pipe inclinations. The temporal variation of the film thickness was measured simultaneously across the entire pipe circumference using a multi-range conductance sensor covering about 7 cm in the axial direction. The recording of the temporal and spatial distribution of the film thickness allows investigation of characteristic interfacial wave parameters such as mean film thickness, frequency, wave height distribution, wave propagation velocities, wave length, and more. The interfacial waves observed across the pipe periphery vary with pipe inclination angle, liquid flow rate and circumferential position. These waves were categorized as ripples, disturbance waves or rogue waves, based on parameters such as frequency, wave height and exceedance. The distribution and the total mass transported by the shear driven waves is also examined. For a low liquid flow rate, a transition from stratified wavy to annular flow was documented with increase in pipe inclination. This data is crucial for better understanding and modeling of evolution of annular and stratified flow regimes as a function of gas and liquid flow rates and pipe inclination.

4:40PM G34.00005 Liquid-liquid phase separation in sessile drops induced by evaporation, HOSEIN SADAFI, Université Libre de Bruxelles, RAMIN RABBIANI, University of Lige, SAM DEHAECK, Université Libre de Bruxelles, TIPS LABORATORY TEAM, THERMODYNAMICS OF IRREVERSIBLE PHENOMENA TEAM — The interplay between two phase change mechanisms of evaporation and liquid-liquid phase separation (demixing) in binary sessile drops of partially miscible liquids is investigated. To determine the onset of the demixing phenomenon, a simple model is developed, which predicts a considerable temperature reduction in the mixture due to evaporative cooling. Temperature reduction alongside with the change of composition lead to demixing in the mixtures. Five stages of the process are identified and explained. For the cases studied here, once the demixing begins through nucleation, a growing fingering pattern is formed at the contact line. The length of the fingers is a function of the initial concentration of the low volatile component. Moreover, the final area of deposition increases with the initial concentration. Experimental tests were performed using a double telescopentric lens.

5:05PM G34.00008 Turbulent Faraday instabilities at the interface between miscible fluids, LOUIS GOSTIAUX, Univ Lyon, Ecole Centrale de Lyon, INSA Lyon, Université Claude Bernard Lyon I, CNRS, LMFA, ANTOINE BRIARD, CMLA, ENS Cachan, CNRS Université Paris-Saclay, BENOT-JOSEPH GRA, DIF, DAM, CEA Arpajon, France — We studied turbulent mixing between two miscible fluids of different densities occurring when the stable interface between the denser and lighter fluids is periodically accelerated. Vertical oscillations trigger the well known Faraday instability, that degenerates into turbulence in the case of a miscible interface. Very few studies have been dedicated to the thickening and saturation of this turbulent mixing zone, and to the associated mass transfers. We realized a large set of idealized experiments in a large container (72x12x90cm), where a two layer system of fresh and salty water was subject to sinusoidal vertical oscillations. Motion was controlled by an hexapod providing vertical accelerations up to 0.7g. We could predict, for a given initial concentration of the low volatile component. Moreover, the final area of deposition increases with the initial concentration. Experimental tests were performed using a double telescopentric lens.

5:06PM G34.00007 Two-phase flow instability at a channel outlet, PAUL R. KANEELIL, AMIR A. PAHLAVAN, KENALPHA KIPYEGON, Princeton University, KRISTEN LEROY, KELSEY STENGEL, SAMUEL WARNER, ANNA M. GALEA, Lung Biotechnology PBC, HOWARD A. STONE, Princeton University. — Parallel flow of two immiscible liquids in a microfluidic channel has a variety of applications including solvent extraction, membrane fabrication, chemical processing, and biomedical design. Here, we investigate a system where two immiscible liquids with the same viscosity enter a channel via two inlets, come into contact and form an interface as they flow side-by-side, and subsequently exit the channel via two outlets. We observe an instability at the exit junction, where the interface periodically oscillates and leads to droplet shedding. By systematically exploring the influence of geometric features and fluid flow on the instability, we characterize the underlying mechanism of the instability and offer pathways to control and suppress it.

¹The authors gratefully thank financial support from European Space Agency (ESA) and the Belgian Federal Science Policy Office (BELSPO) through PRODEX and IAP 7/38 MAST contracts and Fonds de la Recherche Scientifique F.N.R.S. (PDR - DITRASOL contract T.0123.16 and Research Director position of PC).
Taylor dispersion in the presence of cross flow and interfacial mass transfer. TIRAS Y. LIN, ERIC S.G. SHAQFEH, Stanford University — Transverse velocity gradients can enhance the effective diffusion coefficient of a scalar in the primary flow direction, a phenomenon known colloquially as Taylor dispersion. In this work, we perform Taylor dispersion analysis on a pressure-driven flow in a channel with a cross flow, using both perturbation theory and Brownian dynamics simulations. Moreover, we illustrate how mass transfer at the wall affects the evolution of the scalar. We elucidate how the effective diffusion coefficients, effective advective velocities, and effective mass-transfer rates depend on the strength of the cross flow and the wall transfer coefficient, and we perform an asymptotic analysis to investigate the limit of strong cross flow. We discuss a few applications where our results may be useful. For example, in the treatment of a cancerous tumor using nanoparticles, interactions with red blood cells drive nanoparticles in the transverse direction toward the porous blood vessel wall, where they can then be transferred through the wall. Additionally, it has recently been shown that applied forces can cause particles to drift laterally in a viscoelastic channel flow. In both of these applications, our results can be used to understand the resulting particle dispersion.

Dispersion control in deformable microchannels. GARAM LEE, ABIGAIL TAYLOR, Brown University, ALAN LUNER, JEREMY MARZUOLA, University of North Carolina at Chapel Hill, DANIEL HARRIS, Brown University — In fully-developed pressure-driven flow, the spreading of a dissolved solute is enhanced in the flow direction due to transverse velocity variations in a phenomenon now commonly referred to as Taylor dispersion. It is well understood that the characteristics of the dispersion are sensitive to the channel’s cross-sectional geometry. Here we demonstrate a method for manipulation of dispersion properties in a single microchannel via controlled deformation of one of the channel’s walls. Using a rapid-prototyped multi-layer microchip, the upper channel wall is deformed by an external pressure source allowing us to characterize the dependence of the dispersion on the deflection of the channel wall and overall channel aspect ratio. Our experimental measurements are compared directly with theoretical predictions.

Identification and decomposition of advective transfer on microtextured surfaces. YANKING WANG, New Mexico State University, TIE WEI, New Mexico Institute of Mining and Technology, WANG’S CFD LAB COLLABORATION, WEI’S CFD LAB COLLABORATION — Through well-designed physical models and high-fidelity numerical simulations, the physics of advective heat and mass transfer on a solid surface embedded with structured micro pillars is revealed. Two types of advection mechanism are identified, local advection and long-range advection, corresponding to the eddy recirculation in the gaps between streamwise neighboring pillars and the fluid circulation between the regimes above and below pillar tips. The flow topologies suggest that the flow of local advection recirculates primarily in xz plane, and the flow of long-range advection recirculates in yz plane. This fact allows for the decomposition of the two mechanisms by averaging the flow quantities in the streamwise and spanwise coordinates. The enhancement of heat and mass transfer by local and long-range advection is examined for typical geometries, and the transition of the primary transfer method between the two mechanisms due to geometrical change is analyzed in detail. The dependence of local and long-range advection on streamwise and spanwise distances between micro pillars is identified, and this offers promising potential for the accurate control of heat and mass flux in micro fluids.

A Quantitative Study of the Effect of Flow on the Photopolymerization of Fibers. MALCOLM SLUTZKY, HOWARD STONE, JANINE NUNES, Princeton University — The gelation resulting from the interaction between a continuously flowing photo-crosslinkable fluid and pulsed-UV light can be used to produce uniform flexible microfibers. We study this process of fiber production by investigating the conditions required for gelation and by developing a steady-state flow model of the gelation process, which captures the effects of UV exposure on the spatial concentration of radical species and molecular oxygen in the direction of flow. Using this model, we are able to predict critical conditions for fiber production and verify these predictions with our experimentally-observed results. Additionally, we define three regimes of fiber production (in which no fibers, non-uniform fibers, and uniform fibers are produced), qualitatively characterize relationships between fiber length and rigidity, and, with insight drawn from the mathematical model, develop guidelines for the standardized production of uniform fibers with predictable and controllable length.

Measurement of pressure field in microchannel flow from velocity data obtained from micro-PIV. SHINGO OTA, Graduate school of Mechanical Engineering, Tokyo University of Science, Japan, KEN YAMAMOTO, MASAHIRO MOTOSUKE, Department of Mechanical Engineering, Tokyo University of Science, Japan — Dynamics inside and outside of cells and bubbles/droplets in microchannels, e.g., mechanical characteristics of cells and interfacial behavior in multiphase flows, are complicated and difficult to be reproduced by the numerical simulation. To understand these phenomena, a measurement technique that can obtain precise pressure fields of liquid is required. A pressure-field calculation from velocity fields is one possible scenario to obtain the pressure field in microscopic domain where inserting pressure probes is hardly achieved. The present study investigates a reconstruction of pressure fields from velocity fields obtained by micro-PIV based on the Navier-Stokes equation. Fast Fourier transformation (FFT) is used in the pressure-field calculation. And the pressure fields calculated from micro-PIV are compared with CFD results. By preparing artificial particle images with different image resolution, we investigate effects of the image resolution and errors due to the PIV analysis on the accuracy of the pressure-field calculation. As a result, it is shown that the method can reproduce the pressure fields despite the fact that the velocity data contains error. Moreover, the accuracy of both the velocity and the pressure fields can be improved as the increase of the resolution of the artificial particle images.
4:53PM G35.00006 Gas-assisted Taylor Cone-Jets1, FRANCISCO CRUZ-MAZO, Univ. Sevilla (Spain), MAX O. WIEDORN, Center for Free-Electron Laser Science, DESY (Germany), MIGUEL A. HERRADA, Univ. Sevilla (Spain), GISEL E. PENEA-MURILLO, JURAJ KNOSKA, Center for Free-Electron Laser Science, DESY (Germany), SASA BAJT, Deutsches Elektronen-Synchrotron (Germany), HENRY N. CHAPMAN, Center for Free-Electron Laser Science, DESY (Germany), ALFONSO M. GANAN-CALVO, Univ. Sevilla (Spain) — We introduce a way to produce steady micro/nano-liquid jets via electric fields together with co-flowing gas streams. We study the dripping-jetting transition of this configuration theoretically through a global stability analysis as a function of the governing parameters involved. Indeed, we derive two coupled scaling laws that predict both the minimum jet diameter and its maximum velocity. The theoretical prediction provides a single curve that describes not only the numerical computations but also experimental data from the literature for cone-jets. Additionally, we performed a set of experiments to verify what parameters influence the jet length. Due to the diameters below 1 micrometer and high speeds attainable in excess of 100 m/s, this concept has the potential to be utilized for structural biology analyses with X-ray free-electron lasers at megahertz repetition rates as well as other applications.

1Funded by the Ministerio de Economía y Competitividad (Spain) through the Plan Estatal 2013-2016 Retos, and the German Science Foundation (DFG) through the Gottfried Wilhelm Leibniz programme.

5:06PM G35.00007 An Elastic filament in a time-periodic linear shear flow , VIPIN AGRAWAL, DHIRUDABIDITYA MITRA, NORDITA — We numerically study the dynamics of a free elastic filament in a highly viscous linear shear flow in the absence of inertia and brownian motion. We use a bead-spring model with Rotne-Prager viscosity [1]. In time-independent shear flow the filament shows tumbling, C-buckling, and snake-turn, before becoming straight at late times [2,3,4], for different elasto-viscous numbers, Γ – Dimensionless ratio of viscous and elastic stress. In time-periodic flow (Period T), for Γ < Γ1, as expected the filament comes back to its initial position after one period. Surprisingly, for Γ1 < Γ < Γ2, we find two-cycle – the filament comes back to it’s initial shape not after one but two periods. For Γ > Γ2, we observe complex dynamical behaviors. Our results are independent of choice of initial conditions.


5:19PM G35.00008 Flow physics of single-phase laminar flow through diamond microchannel , SANDEEP GOLI, SANDIP KUMAR SAHA, AMIT AGRAWAL, Indian Institute of Technology, Bombay — Diamond microchannel is a varying cross-section microchannel with diverging and converging flow passages. Flow in such devices has significance in the design of micromixers, micropumps and microreactors. Three-dimensional numerical analysis of single-phase laminar liquid flow has been performed to understand the effect of the given configuration on flow parameters such as pressure drop and Poiseuille number. The results show that pressure drop in given configuration is consistent with theoretical predictions, which suggests that existing correlations for uniform microchannels can be applied to the present configuration. Towards this, an appropriate length scale has been identified to make the hydrodynamic flow resistance of the diamond microchannel is same as that of an equivalent uniform microchannel. This is located at 1/7th of the total length of the microchannel from its inlet. This location makes the hydrodynamic resistance of microchannel independent of its geometric and flow parameters. In addition, flow physics has been studied with the help of velocity, pressure and shear stress profiles.


3:48PM G36.00001 Time Periodic Electroosmotic Flow in Cylindrical Microchannel with Heterogeneous Surface Charge , AMINUL KHAN, PRASHANTA DUTTA, Washington State University — Mixing of species is very slow in microfluidic due to creeping flow. Time periodic electroosmotic flow (EOF) has been used for faster mixing in microdevice by rapid stretching and folding of fluid streams. Furthermore, surface inhomogeneities are also explored to expedite the mixing process. Although several analytical models exist for each individual case, there is no analytical solution for time-periodic EOF in a heterogeneously charged microchannel. In this work, a general analytical model has been developed for time-periodic EOF through cylindrical microchannel by solving Navier-Stokes equation with slip velocity conditions at the channel wall. Results show that the axial variation of surface charge yields diverse flow patterns containing counter-rotating vortices. The extent and strength of vortices are characterized by channel size, charge distribution and the period of electric field. As the electric field frequency or channel diameter increases, vortices are shifted towards the channel surface and the perturbed flow region confined near the channel wall. Also, the number of vorticities depends on the periodicity of the surface charge. Our analytical model can be used for effective micromixer design by manipulating the surface charge pattern.

4:01PM G36.00002 Particle trapping in merging flow junctions by fluid-solute-colloid-boundary interactions1, SANGWOO SHIN, University of Hawaii at Manoa, JESSE AULT, Oak Ridge National Laboratory, AMY SHEN, Ōkinawa Institute of Science and Technology — Merging of different streams in channel junctions represents a common mixing process that occurs in systems ranging from soda fountains and bathtub faucets to chemical plants and microfluidic devices. Here, we report a sudden trapping of colloidal particles in a merging flow junction when the merging streams have a salinity contrast. We show experimentally and numerically that the particle trapping is a consequence of complex interactions between diffusionosmosis, diffusiophoresis, and the freestream flow. A delicate balance of these transport processes results in a stable vortex near the junction that traps the particles in various modes depending on the flow conditions. We use 3-D particle visualization and numerical simulations to provide a rigorous understanding of the observed particle trapping phenomenon. The trapping mechanism we identify is unique from the well-known inertial trapping that is enabled by the vortex breakup. As the current particle trapping can occur even at Reynolds number below unity. Our study demonstrates a good example of how nonlinear, coupled fluid-solute-colloid-boundary dynamics can result in peculiar particle behavior in simple flow systems.

1This material is based upon work supported by the National Science Foundation under Grant No. CBET-1930691.
4:14PM G36.00003 Non-intuitive behavior in concentrated suspension of ideally polarizable particles in an electric field1. SIAMAK MIREFENDERESKI, JAE SUNG PARK, University of Nebraska Lincoln — Large-scale numerical simulations are used to analyze the dynamics of ideally polarizable particles in concentrated suspensions under the effects of nonlinear electrokinetic phenomena. Particles are assumed to carry no net charge and considered to undergo the combination of dielectrophoresis and induced-charge electrophoresis termed dipolophoresis. The suspension dynamics seems to be hindered up to semi-dilute regimes by the increase in the magnitude of excluded volume interactions. Interestingly, a non-intuitive suspension behavior is observed in concentrated regimes, where the hydrodynamic diffusivity starts to increase with volume fraction and reach a local maximum before decreasing as approaching random close packing. This behavior is rationalized through an examination of the velocity fluctuations, suspension microstructure, and number-density fluctuations. We conclude that the non-intuitive behavior is attributed to a consequence of particle contacts, depending on the dominant mechanism of particle paring. While contacts are expected to occur along the field direction in dilute or semi-dilute regimes, very strong and massive contacts along the direction perpendicular to the applied field arise, promoting the non-intuitive behavior observed in concentrated regimes.

1The authors gratefully acknowledge the financial support from the Collaboration Initiative and Interdisciplinary Research Grants at the University of Nebraska

4:27PM G36.00004 Diffusiophoresis in Multivalent Electrolytes1. JESSICA L. WILSON, SUIN SHIM, ANKUR GUPTA, HOWARD A. STONE, Princeton University — Diffusiophoresis is the spontaneous movement of colloidal particles in a concentration gradient of ions. As a small-scale phenomenon that harnesses energy from concentration gradients, diffusiophoresis may prove useful for passively manipulating particles in lab-on-a-chip type applications. Though naturally occurring ions are often multivalent, experimental studies on diffusiophoresis have been mostly limited to monovalent electrolytes. In this work, we investigate the motion of negatively charged polystyrene particles in one-dimensional salt gradients for a variety of multivalent electrolytes. Our results indicate that the valence combination of cation and anions significantly impacts the diffusiophoretic mobility of the particles. In addition, the ion valence also modifies the ambipolar diffusivity, which in turn influences the motion of the particles by changing the timescale at which the concentration gradients evolve. We also develop a 1D model and obtain a good agreement between our experimental and modeling results. Our results are applicable to systems where the chemical concentration gradient is made up of multivalent ions, and motivate future research to manipulate particles by exploiting ion valence.

1NSF CBET - 1702693

4:40PM G36.00005 Electrophoresis of Colloids via Asymmetric Rectified Electric Fields. TIMOTHY HUI, JOSHUA LAU, S.M.H. HASHEMI AMREI, GREGORY MILLER, WILLIAM RISTENPART, University of California Davis — Charged particles suspended in a dilute electrolyte solution exhibit complex aggregation and levitation behaviors in response to applied oscillatory fields. In particular, an experimentally observed bifurcation in the particle height over an electrode was recently shown to be qualitatively consistent with a force balance between gravity and an electrophoretic force due to the Asymmetric Rectified Electric Fields (AREF) that occur in electrolytes with unequal mobilities. Here, we elaborate on the dynamics of particle electrophoresis in response to AREFs. Using a combination of optical and confocal fluorescence microscopy, we measure the colloidal particle levitation and aggregation dynamics as a function of the applied field properties and electrolyte composition. We show that the dynamics are broadly consistent with numerical calculations of the AREF driving force, and we discuss the implications for precise control and electrophoretic trapping of particles using oscillatory fields.

4:53PM G36.00006 Nonlinear concentration waves in current monitoring method for measurement of electroosmotic flow. MOHAMMAD BABAR, KAUSHLENDRA DUBEY, SUPREET SINGH BAHGA, Department of Mechanical Engineering, Indian Institute of Technology Delhi — Current monitoring method for measurement of electroosmotic flow (EOF) in a microchannel involves displacement of a binary electrolyte initially filled in the channel by a similar electrolyte but having a different conductivity, under the effect of electric field. A fixed voltage is applied across the channel ends and the temporal change in current is measured. Because a conductivity gradient in a binary electrolyte migrates only due to EOF and not due to electromigration, the time taken by one electrolyte to displace another gives an estimate of EOF. This displacement time is usually independent of whether a high-conductivity electrolyte displaces a low-conductivity electrolyte or vice versa. However, few studies have reported that, for low-conductivity electrolytes, the displacement time depends on whether the displacing electrolyte has higher or lower conductivity than the displaced electrolyte. In this study, we show that this directional dependence of displacement time is a result nonlinear concentration waves, such as shock and rarefaction waves, that form when surface conduction is comparable with bulk conduction. We present analytical expressions for current-time relationship in this regime and validate them with experimental observations.

5:06PM G36.00007 Temperature-gradient induced enhanced load bearing capacity of lubricated systems. SIDDHARTHA MUKHERJEE, Advanced Technology Development Center, Indian Institute of Technology Kharagpur, Kharagpur, India-721302, SUNANDO DASGUPTA, Department of Chemical Engineering, Indian Institute of Technology Kharagpur, Kharagpur, India-721302, SUMAN CHAKRABORTY, Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, Kharagpur, India-721302 — Lubricated systems, which are pretty common in some engineering and biological settings, perform on the basis of their load bearing performances. In recent years, a good volume of research has been directed to strategize the performance enhancement of such devices by coupling interfacial phenomenon like electrokinetics with the substrate deformability. However, the deformation characteristics and subsequent load carrying capacity of such systems in presence of thermal gradient have not been explored. Here, we have unveiled that by exploiting a unique coupling of electrokinetics, externally applied thermal gradient, hydrodynamics and substrate flexibility, one can modulate the load bearing capacity of a planar slider bearing considerably. Our studies reveal that the alteration in the electrical potential distribution upon application of temperature gradient and its influence on the resulting hydrodynamics, coupled with substrate compliance, may give rise to significant enhancement in load capacity. We envisage that this analysis may construct a new window in the context of improved design of lubricated systems where the interplay between thermal, electrokinetic and hydrodynamic aspects can be coupled together.
In laminar pipe flow — a linear-onset regime, a time-periodic non-linear regime with the generation of vortices qualitatively similar to the von Kármán vortex street, and eventually a coherent regime which leads to the mixing of the two fluids. A linear stability analysis reveals the occurrence of five finite wavenumber modes of instabilities. The electro-hydrodynamic instabilities originating upon application of electric field stimulate a pair of shorter-wavelength electric field modes E-I and E-II beyond a critical value of electric Rayleigh number. For higher viscosity difference between the fluids, the relative longer wavelength viscous mode (V-mode) appears. Beyond a critical Schmidt number, a diffusive mode (D-mode) appears, which is qualitatively similar to the interfacial instabilities of the immiscible fluids. The K-mode of instability appears due to contrast in ionic mobility values. The reported phenomenon can be harnessed for microscale mixing, pumping, heat and mass transfer, and reaction engineering.

This research was supported by the National Science Foundation (Award 1538090).

1DST SERB, Grant no. EMR/2016/001824, Government of India
2Chemical Engineering, IIT Guwahati, India
3Chemical Engineering, IIT Guwahati, India
4Chemical Engineering, IIT Guwahati, India

Sunday, November 24, 2019 3:48PM - 5:32PM –
Session G37 Particle Laden Flows  619 -  Ellen Longmire, University of Minnesota, Twin Cities

4:01PM G37.00002 Effect of turbulent puffs on the Segré-Silberberg effect in particulate pipe flow1, SAGNIK PAUL, ELLEN LONGMIRe, University of Minnesota — Particles in laminar pipe flow tend to accumulate at a certain radius close to the wall. When the flow becomes unstable such that turbulent puffs occur, the puffs can disturb the accumulation of particles. In our current work, we investigate the behavior of polystyrene beads in a neutrally buoyant liquid (μ =1046 kg-m-3) over pipe Reynolds numbers in the laminar and transitional range. In our setup, we observe discrete puffs between Re=2100 and 2600 and study the distribution of particles versus radius. The transition Reynolds number and particle distribution are expected to be dependent on the ratio of the particle diameter (d) to pipe diameter (D), and the particle volume fraction (φ). Particle distributions and motions are observed in planes aligned parallel and perpendicular to the pipe axis using both LED and laser sheet imaging. Images are analyzed to quantify particle position distributions. In laminar flow with D/d=43 and φ =0.005, the maximum concentration of particles occurs at 0.44D. In transitional flow at higher Re, the puffs exhibit strong disturbances near the walls that disrupt the local particle accumulation. The effect and relative importance of these disturbances on particle concentration and velocity will be discussed.

1Supported by NSF(CBET 1605719).

4:14PM G37.00003 Experimental investigation of particle-laden under-expanded jets1, TAEHOON KIM, RUI NI, Johns Hopkins University, JESSE CAPECELATRO, YUAN YAO, GREGORY SHALLCROSS, University of Michigan, MANISH MEHTA, NASA Marshall Space Flight Center, JASON RABINOVITCH, NASA Jet Propulsion Laboratory — We present an experimental investigation of particle dynamics in sonic under-expanded jets. In this study, the mass loading and particle slip velocity are independently controlled by using a particle injector and a particle accelerator, which are integrated together in a hopper-style particle feeding system. An ultra-high-speed Schlieren imaging and a particle tracking system operating at 2-5 million fps were employed to acquire the dynamics of both the gas phase and particle phase. Statistics for both particle-free and particle-laden under-expanded jets in the near field will be discussed. These new results will provide a rich dataset for expanding our knowledge in compressible multiphase flow problems as well as validating models and simulations in this regime.

1NASA FY19 MSFC Center Innovation Fund

4:27PM G37.00004 Polydisperse Droplet Size Growth in a Turbulent Air Flow - Effects of Droplet Number Density and Size , SHYAM KUMAR M, PhD Scholar, Department of Aerospace Engineering, Indian Institute of Technology Madras, Chennai 600036, India, S.R CHAKRAVARTHY, MANIKANDAN MATHUR, Professor, Department of Aerospace Engineering, Indian Institute of Technology Madras, Chennai 600036, India — Interaction of polydisperse droplets in a turbulent air flow features prominently in a wide range of phenomena. Here, we present an experimental study on the collective effects of droplet number density (ND) and mean droplet size ($D_{10}$) on the droplet size growth. For each (ND, $D_{10}$), we observe an optimum turbulent intensity and a corresponding maximum droplet size growth rate $R^*$. Interestingly, an increase (decrease) in ND($D_{10}$) doesn’t influence $R^*$ up to some threshold values of ND and $D_{10}$, beyond which a sudden increase in $R^*$ is observed. The observed trend is understood in terms of droplet pair dispersion, as observed from Long Distance Microscopy images. The dispersal of droplet pairs is negligible till the threshold conditions are reached. Surprisingly, a substantial increase in dispersion is observed beyond the threshold values, which potentially increases the collision probability, and hence $R^*$. The opposing effects of ND and $D_{10}$ on collision rates cause a negligible variation of $R^*$ up to the threshold values; however, the effect of ND overrides the effect of $D_{10}$ beyond the threshold values, and hence causes a sudden increase in $R^*$. 

1Supported by NSF(CBET 1605719).
4:40 PM G37.00005 Multi-scale characterization of the effect of gas swirl on two-fluid coaxial atomization. NATHANIEL MACHICOANE, PETER D. HUCK, University of Washington, TIMOTHY B. MORGAN, JULIE K. BOTHELL, Iowa State University, RODRIGO OSUNA-OROZCO, University of Washington, DANYU LI, THEODORE J. HEINDEL, Iowa State University, ALAN L. KASTENGREN, Argonne National Laboratory, ALBERTO ALISEDA, University of Washington — This work aims at developing a better mechanistic understanding of the processes that control droplet formation and transport in coaxial two-fluid atomization. The goal is to experimentally develop spray control strategies and implement feedback control of the spray characteristics. We will present results on the impact of modulation of the swirl ratio (ratio of gas flow rate with tangential momentum to the total gas flow rate) on the physics of the atomization, from the formation of liquid ligaments close to the nozzle, to droplet size and spatiotemporal distributions in the mid-field. This parameter dominates the spray structure when it is above a critical value and its modulation in time has a non-linear effect on the spray dynamics that can be used to shape it towards a desired state (the control goal). Synchrotron X-ray measurements provide detailed information of the spray near-field, while time-resolved optical measurements of the spray structure, as well as the droplet size and velocity, are collected in the mid-field. The response of the spray to the open-loop actuations is captured from these measurements. This is used to develop reduced order model for feedback control and to validate assumptions used for computational, adjoint-based, control strategies.

4:53 PM G37.00006 Dynamics of turbidity currents: Ambient fluid entrainment and basal drag1 JORGE SALINAS, University of Florida, MRUGESH SHRINGARPURE, ExxonMobil Upstream Research Company, MARIANO CANTERO, Instituto Balseiro, National Council of Scientific and Technical Research (CONICET), S. BALACHANDAR, University of Florida — Turbidity currents are sediment laden flows that run along inclined or horizontal surfaces. They are driven by the excess hydrostatic pressure resulted from the density difference between the current carrying sediment and the clear ambient fluid. The amount of particles carried is influenced by the strong coupling between turbulence and suspended sediment. In this work we perform direct numerical simulations (DNS) of spatially evolving, spanwise- homogeneous turbidity currents. We focus our attention on the process of entrainment of clear ambient fluid that happens at the interface between the current and the ambient layer. Moreover, the study of basal drag is performed. Turbidity currents have been studied in the past by means of layer-averaged equation models, which make use of several closure models. Two of these models are the entrainment assumption and the basal drag model. In this work we analyze the dependence of these models with the flow parameters (bulk Richardson and Reynolds numbers) together with the settling velocity of the sediment. We pay attention to the regime where the bulk properties of the flow vary slowly, called normal condition. Furthermore, we analyze the effect of acceleration/deceleration of the flow away from normal condition.

1 The authors thank ExxonMobil for their support

5:06 PM G37.00007 Particle-laden upward jet in a crossflow: particle dispersion and tracking of particle source1 JOOYEON PARK, HYUNGMIN PARK, Seoul National University — We experimentally investigate the particle dispersion due to the vortical interaction in the particle-laden upward jet with a crossflow focusing on a large scale phenomena in multiple planes. We vary the velocity ratio (R) between the jet and crossflow, which is classified into three regimes of no crossflow, 3.0 – 27.42. For each case, the air flow and particle velocities are measured by PIV, and the particle distribution is obtained by planar nephelometry. For lower R, due to stronger counter-rotating vortex pairs (CVPs) in a continuous phase, drag force on particles becomes dominant so that the particles are swept from the jet center near the exit for St << 1, but for St >> 1, the particles tend to travel along the jet center regardless of vortical effects. Interestingly, only for St = 1 (irrespective of R), the particles agglomerate along the jet center before the CVF collapses. Finally, based on these observations, a 3D dispersion model is developed, which is used for the estimation of particle source location and validated with the experimental data.

1 Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

5:19 PM G37.00008 Physical origins of the dependence of drag force on density ratio through fully-resolved direct numerical simulation of particle-laden to bubbly flow1 VAHID TAVANASHAD, SHANKAR SUBRAMANIAM, Iowa State University — The objective of this study is to understand the dynamics of freely evolving particle suspensions over a wide range of particle to fluid density ratios, from solid particles in a gas (high density ratio) to bubbles in a liquid (low density ratio). The dynamics of particle suspensions are characterized by the average momentum equation, where the dominant contribution to average interphase momentum transfer is the mean drag force. The mean drag is characterized using fully-resolved simulation over a wide range of density ratios in a canonical problem: a statistically homogeneous suspension where a steady mean slip velocity between the phases is established by an imposed mean pressure gradient. We explain the change of mean drag of freely evolving particle suspensions with reference to a fixed bed by considering the emergence of spatial structure in the particle configuration, effect of particle velocity fluctuations, and the mobility of particles. These considerations could be used to develop a physics-based drag law for dispersed multiphase flows. The study of the fluctuations of the drag force on individual particles about the mean drag shows that the force distribution follows a normal distribution with a variance from the mean drag that decreases with decreasing density ratio.

1 This material is based upon work supported by the National Science Foundation under Grant No. 1438143.


3:48PM G38.00001 Heat transfer and flow structures of Rayleigh-Benard convection in regular porous media1 SHUANG LIU, LINFENG JIANG, Tsinghua University, KAI LEONG CHONG, University of Twente, XIAOJUÉ ZHU, Harvard University, ZHENGANG WANG, University of Science and Technology of China, ROBERTO VERZICCO, University of Rome Tor Vergata, RICHARD STEVENS, DETLEF LOHSE, University of Twente, CHAO SUN, Tsinghua University — We report on a numerical study of porous media Rayleigh-Benard (RB) convection for the Rayleigh number Ra from 105 to 110 at various porosities 6. The porous media is constructed by an array of circular, solid obstacles located on a square lattice. For a given 6 two flow regimes are identified with different heat transport properties and flow structures. In the small-Ra regime the heat transport is reduced compared with the classical RB convection without obstacles and the flow is dominated by coherent thermal plumes, while in the large-Ra regime the heat transport is enhanced and the flow is dominated by fragmented plumes. The Nusselt number follows different scaling behaviors in the two regimes, and the regime crossover occurs when the thermal boundary layer thickness is comparable to the particle separation.

1 This work is supported by the Natural Science Foundation of China under grant nos. 91852202 and 11672156.
Mohammad Zargartalebi, Anne Benneker, University of Calgary — Local overheating transfer in microchannel from full- to fluid-dominated convection. We find good agreement between critical depth ratios predicted from our theory and actual values.

Critical Rayleigh numbers of the respective uncoupled domains, we propose a theory for predicting the depth ratio required for the transition sub-soil reservoirs, etc.), being able to predict the critical depth ratio where this convection shift occurs is particularly timely. Based on the in numerous projects of industrial, environmental, and geophysical importance (oil recovery, carbon dioxide sequestration, contamination in sub-soil reservoirs, etc.), being able to predict the critical depth ratio where this convection shift occurs is particularly timely. Based on the critical Rayleigh numbers of the respective uncoupled domains, we propose a theory for predicting the depth ratio required for the transition from full- to fluid-dominated convection. We find good agreement between critical depth ratios predicted from our theory and actual values, especially in the small Darcy number limit.

Matthew McCurdy, Florida State University — In superposed fluid-porous media systems, the ratio of the fluid height to the porous medium height exerts a significant influence on the behavior of the coupled system, most notably with its impact on resulting convection cells. Altering the depth ratio slightly can trigger a transition from full-convection where convection cells extend throughout the entire domain to fluid-dominated convection where cells occupy only the fluid region. With current interest surrounding superposed fluid-porous medium systems in numerous projects of industrial, environmental, and geophysical importance (oil recovery, carbon dioxide sequestration, contamination in sub-soil reservoirs, etc.), being able to predict the critical depth ratio where this convection shift occurs is particularly timely. Based on the critical Rayleigh numbers of the respective uncoupled domains, we propose a theory for predicting the depth ratio required for the transition from full- to fluid-dominated convection. We find good agreement between critical depth ratios predicted from our theory and actual values, especially in the small Darcy number limit.

4:14PM G38.00003 Transition of convection in coupled fluid-porous media systems

Mohammad Zargartalebi, Anne Benneker, University of Calgary — In superposed fluid-porous media systems, the ratio of the fluid height to the porous medium height exerts a significant influence on the behavior of the coupled system, most notably with its impact on resulting convection cells. Altering the depth ratio slightly can trigger a transition from full-convection where convection cells extend throughout the entire domain to fluid-dominated convection where cells occupy only the fluid region. With current interest surrounding superposed fluid-porous medium systems in numerous projects of industrial, environmental, and geophysical importance (oil recovery, carbon dioxide sequestration, contamination in sub-soil reservoirs, etc.), being able to predict the critical depth ratio where this convection shift occurs is particularly timely. Based on the critical Rayleigh numbers of the respective uncoupled domains, we propose a theory for predicting the depth ratio required for the transition from full- to fluid-dominated convection. We find good agreement between critical depth ratios predicted from our theory and actual values, especially in the small Darcy number limit.

4:27PM G38.00004 Optimization of Porous Medium Structure to Enhance Heat Transfer in Microchannel

Mohammad Zargartalebi, Anne Benneker, University of Calgary — Local overheating in superposed fluid-porous media systems, the ratio of the fluid height to the porous medium height exerts a significant influence on the behavior of the coupled system, most notably with its impact on resulting convection cells. Altering the depth ratio slightly can trigger a transition from full-convection where convection cells extend throughout the entire domain to fluid-dominated convection where cells occupy only the fluid region. With current interest surrounding superposed fluid-porous medium systems in numerous projects of industrial, environmental, and geophysical importance (oil recovery, carbon dioxide sequestration, contamination in sub-soil reservoirs, etc.), being able to predict the critical depth ratio where this convection shift occurs is particularly timely. Based on the critical Rayleigh numbers of the respective uncoupled domains, we propose a theory for predicting the depth ratio required for the transition from full- to fluid-dominated convection. We find good agreement between critical depth ratios predicted from our theory and actual values, especially in the small Darcy number limit.

4:40PM G38.00005 Experimental investigation on the scaling of convective dissolution in porous media

Marco De Paoli, Mobin Alipour, Alfredo Soldati, Institute of Fluid Mechanics and Heat Transfer, TU Wien — Porous media convection in Rayleigh-Bénard-type configuration is of paramount importance in many industrial and environmental applications. However, the fundamental behavior of the dissolution flux and its dependence on the system parameters are not yet well understood. Simulations and experiments give opposite indications. In particular, the results of two-dimensional Darcy simulations suggest that the dissolution rate during the convection-dominated regime is constant, whereas Hele-Shaw experiments show that it exhibits a Rayleigh-dependent behavior. With the aid of a novel experimental setup, in which the geometrical properties of the Hele-Shaw cell are varied independently, we obtain accurate measurements of solute fluxes and explain the Rayleigh-dependent character of the dissolution rate observed in previous numerical and experimental studies. Finally, we observe that non-Darcian effects (e.g., mechanical dispersion) may influence the dissolution rate in Hele-Shaw flows and possibly lead to the mismatch between experimental and numerical results.

4:53PM G38.00006 Longitudinal heat conduction effects in a microchannel filled with a porous medium and subjected to a uniform heat flux

Ian Monsivaís, National Autonomous University of Mexico, Jose Lizardi, Autonomous University of Mexico City, Federico Méndez, National Autonomous University of Mexico — The conjugate heat transfer between the walls of a microchannel and a fluid circulating inside is numerically studied. The microchannel is filled with an homogeneous porous medium and subjected to a uniform heat flux on the external walls of the microchannel. The governing equations are written in dimensionless form and basically, we show the existence of two dimensionless parameters that govern the problem: the Darcy number, Da, and the conjugate heat transfer parameter $\alpha_c/\varepsilon^2$. The numerical predictions show that for $\alpha_c/\varepsilon^2 > 1$, the temperature of the fluid at each point of the microchannel is higher than in the case of $\alpha_c/\varepsilon^2 << 1$. These limits are well known as the thermally thin and thermally thick wall limits respectively.

5:06PM G38.00007 Experimental evaluation of thermal transport in partially-porous channel flow

Shilpa Vijay, Mitul Luhar, University of Southern California — While convective heat transfer in channels completely filled with metal foams has been studied extensively, heat transfer in channels that are partially filled with porous foams is less well understood. Previous direct numerical simulations for partially porous channel flow indicate that large vortex structures enhance turbulent heat transfer at the porous medium-unobstructed flow interface. This project aims to experimentally investigate this interfacial thermal transport. The experimental setup involves commercially-available Aluminum foams attached to a heater block and placed in a forced convection arrangement adjacent to an unobstructed channel of equal height. Pressure drop and temperature measurements have been made across the porous section for bulk Reynolds number ranging from 800-3500. Particle Image Velocimetry (PIV) measurements made at a subset of these Reynolds numbers confirm the emergence of interfacial vortex structures in certain conditions. Currently, heat transfer performance in this system is evaluated via estimates of the Nusselt number and friction factor. Ultimately, the PIV dataset will be used to quantify the effect of the interfacial turbulent flow on thermal transport.
Simultaneous measurement of velocity and concentration fields in Hele-Shaw cell. MOBIN ALIPOUR, TU Wien, University of Udine, DR. MARCO DE PAOLI, TU Wien, ALFREDO SOLDATI, TU Wien, University of Udine — Convective dissolution in porous media has a wide range of applications such as sea ice formations, evaporation from soil and geological carbon dioxide storage. While the wealth of computational studies has shed a light on some fundamental features of this flow, experimental techniques currently adopted do not allow an easy, simultaneous and accurate measurement of concentration and velocity fields and we aim precisely at this gap. In this work, we performed experiments in Hele-Shaw geometry and used an optical technique to obtain the solute concentration field. We propose a concentration-based velocity reconstruction (CVR) algorithm, i.e. a new method to reconstruct the velocity field from the solute concentration measurements. In particular, measurements of the concentration gradients are used to reconstruct the velocity field from the momentum transport equation. Effect of gap thickness and non-Darcian behavior of the flow are analyzed by means of both concentration and velocity observations of the fingers. We compare the CVR results with the velocity fields obtained via particle tracking velocimetry (PTV) measurements, giving the guidelines for future experimental works.

Sunday, November 24, 2019 3:48PM - 5:32PM
Session G39 Geophysical Fluid Dynamics Ocean I 6a - Jim Thomas, Woods Hole

3:48PM G39.00001 Cross-Shore Structure of Alongshore Flow over a Coral Reef Shelf, GENO PAWLAK, ANDRE AMADOR, Mechanical and Aerospace Engineering, UCSD, ISABELLA ARZENO, SARAH GIDDINGS, MARK MERRIFIELD, Scripps Institution of Oceanography, UCSD — Tidally driven alongshore flow over a reef shelf on O‘ahu, Hawai‘i is examined using spatial velocity measurements from an autonomous underwater vehicle (AUV) along with time series observations of the alongshore pressure gradient. Depth-averaged velocities are reconstructed from AUV-based velocity observations as a function of cross-shore distance assuming tidal periodicity. Ensemble averages of the alongshore pressure gradient and velocities versus tidal phase from multiple AUV surveys reveal characteristics akin to an oscillatory boundary layer, with the nearshore flow leading the offshore flow and with a corresponding attenuation in velocity magnitude in shallower regions. Analysis of the depth-averaged alongshore momentum balance indicates that the cross-shore structure and evolution of the tidal boundary layer is strongly modulated by a balance between local acceleration, barotropic pressure gradient, and bottom drag. This primary balance allows estimation of the drag coefficient over depths spanning from 25 to 5 m. Estimates agree with analysis of time series data and compare favorably with drag coefficients estimated from AUV-based roughness mapping. Roughness data suggest that larger scales, at wavelengths comparable to the depth, play a more significant role than smaller meter-scale roughness.

4:01PM G39.00002 Predicting orientation of coastal-zone Langmuir cells influenced by misaligned current, wind and wave forcing, KALYAN SHRESTHA, WILLIAM ANDERSON, UT Dallas — Multi-scale physical processes that involve the interaction between winds, waves, and mean currents regulate turbulence in the upper ocean boundary layer (OBL). Among them, Langmuir turbulence is one such process. In coastal settings, Langmuir turbulence is subjected to additional shear from mean currents. Thus, wind-wave-mean current parameter space and the system response to their disequilibrium becomes an important study for improved parameterizations of OBL processes with wave effects. As such, this investigation considers idealized cases of wind-wave-mean current misalignment and attempts to predict the resultant orientation of coastal Langmuir cells. An a priori predictive model based on mean Lagrangian shear direction is formulated and direct comparisons have been performed successfully with a thorough list of large-eddy simulation results of the Craik-Leibovich equations. The prognostic model as well as the numerical results depict that the resultant cells maintain coherency (although, diminishes with increasing obliquity between the underlying forces) and aligns at an intermediate angle to the range of imposed forces. This was further justified with theoretical developments, which involved analysis of the vorticity transport equation to assess the terms responsible for the sustenance of streamwise vorticity.

4:14PM G39.00003 Study of surface wave effect on the turbulence underneath using wave-phase-resolved simulation, ANQING XUAN, BING-QING DENG, LIAN SHEN, University of Minnesota — The Eulerian orbital motions and the Lagrangian motions of the water surface waves introduce different time scales to the turbulence underneath. In this study, the turbulence dynamics under progressive surface waves are studied using wave-phase-resolved simulations. Compared to the traditional wave-phase-averaged approach of modelling the wave-turbulence interaction, the fast turbulent motions with time scales similar to the waves are directly resolved. Based on the simulation data, we find that the coherent turbulence structures and turbulence statistics are wave-phase dependent. The mechanisms of the wave-phase variation of the turbulence are analyzed in the wave-phase-resolved frame and it is found that the variation is due to the periodic stretching and tilting of the wave orbital straining. The correlations between the wave phase and turbulence statistics, such as the vorticity and Reynolds stresses, are further quantified and their wave-phase-averaged contribution to the wave-turbulence interactions are modelled.

4:27PM G39.00004 Phase-Resolved Ocean Waves Prediction via Machine Learning, FAZLOLHA MOHAGHEGHI, University of California-Los Angeles, MOHAMMAD-REZA ALAM, University of California-Berkeley, JAYTHI MURTHY, University of California-Los Angeles — Phase-resolved prediction of ocean waves is one of the most important outstanding problems in ocean science and engineering. With an accurate prediction of ocean surface height, extreme events such as rogue waves can be braced for, or entirely avoided (e.g. via rerouting of ocean vessels); ocean wave harnessing devices can tune up in real-time to take the most energy out of incoming waves; and the final destination of passive floating particles (e.g. pollutants) can be precisely determined. The problem, however, is very challenging because it uses equations that are not linear; hence even infering wave components that make up a given surface is already a difficult task. Here we show that a Convolutional Recurrent Neural Network (CRNN) has a strong potential to efficiently make real-time prediction of nonlinear non-breaking ocean waves. We use extensive direct simulation of nonlinear ocean waves to train and then test our proposed CRNN-based methodology. Each input node in our CRNN is composed of the discretized surface elevation in a given spatial domain at a specific time. The network takes a time-window of input nodes, and outputs the spatiotemporal prediction of surface waves. Accuracy, reliability and limitations of the proposed methodology are discussed.

4:40PM G39.00005 Machine Learning for Trajectory Prediction in Geophysical Flows1, PHILIP YECKO, Cooper Union, ERIC FORGOSTON, Montclair State University, KEVIN YAO, The Cooper Union — Echo state network based machine learning (ML) is applied to two elementary models of ocean circulation: the well-known double-gyre stream function model with time-variable forcing and a one-layer quasi-geostrophic (QG) basin model. These models are used to generate time-dependent two-dimensional stream function fields, from which flow maps, trajectories and ensembles of trajectories are computed, assuming ideal particles. Varied physical model parameters allow sampling of a wide range of dynamical behaviors corresponding to diverse geophysical flow regimes; the QG PDE model can realize Munk, Stommel or strongly nonlinear time-dependent solutions. We evaluate the effectiveness and fidelity of our machine learning approach in capturing the characteristics of trajectories, both directly and indirectly, via stream function field prediction. We assess the predictive power of ML models against other predictive and descriptive models of QG flows, including finite time Lyapunov exponent, or FTLE, accounting for the role of physical and numerical parameters on our results.

1 partially supported by NSF CMMI awards 1462823 and 1462884
4:53PM G39.00006 Predictability of ROMS-Ocean State Ocean Model using Information Theory

Aakash Sane, Baylor Fox-Kemper, Brown University, David Ullman, Christopher Kincaid, Lewis Rothstein, University of Rhode Island, University of Rhode Island Collaboration, Brown University Collaboration — The Ocean State Ocean Model (OSOM) is an implementation of the Regional Ocean Modeling System (ROMS) covering Rhode Island waterways which includes the Narragansett Bay, Mt. Hope Bay, and nearby regions including the shelf circulation from Long Island to Nantucket. Our focus is on modeling the physical aspects of the Bay in order to build a forecast and prediction system. Perturbed ensemble simulations with altered initial condition parameters (temperature, salinity) are combined with concepts from Information Theory to quantify the predictability of the OSOM forecast system. Predictability provides a theoretical estimate of the potential forecasting capabilities of the model in the form of prediction time scales and enhances readily estimable timescales such as the freshwater flushing timescale. The predictability of the OSOM model is around 10-40 days, varying by perturbation parameters and season.

This material is based upon work supported in part by the National Science Foundation under EPSCoR Cooperative Agreement #OIA-1655221

5:06PM G39.00007 Uncertainty quantification of trajectory clustering in ocean ensemble forecasts

Guilherme Salvador-Vieira, Michael Allshouse, Northeastern University, Irina Rypina, Woods Hole Oceanographic Institution — Identifying coherent structures in unsteady flows helps differentiate flow regions based on material transport. Partitioning flows into regions that minimally mix with their surroundings in the oceans, for instance, can assist search-and-rescue planning by reducing the search domain. One partitioning method is the spectral clustering of trajectories, which maximizes within-cluster similarities while minimizing between-cluster similarities. For ocean applications, however, in addition to the complex dynamics, there are several sources of uncertainty: model initialization and parameters, limited knowledge of the processes, boundary conditions, and forcing terms. Therefore, when applied to ocean forecasting, the clustering method should analyze multiple realizations, identify robust features, and quantify the uncertainty. We present an investigation of the sensitivity of the spectral clustering method to uncertain parametrization and noise through application to simulations of an analytic geostrophic flow model. We then apply this approach to an operational coastal ocean forecast and compare the results with observational drifter data from a field study.

5:19PM G39.00008 Geophysical turbulence dominated by inertia-gravity waves

Jim Thomas, Woods Hole Oceanographic Institution, USA and Dalhousie University, Canada — Recent evidence from both oceanic observations and global-scale ocean model simulations indicate the existence of regions where low-mode internal tidal energy dominates over that of the geostrophic balanced flow. Inspired by these findings, we examine the effect of the first vertical mode inertia-gravity waves on the dynamics of balanced flow using an idealized model obtained by truncating the hydrostatic Boussinesq equations on to the barotropic and the first baroclinic mode. On investigating the wavebalance turbulence phenomenology using freely evolving numerical simulations, we find that the waves continuously transfer energy to the balanced flow in regimes where the balanced-to-wave energy ratio is small, thereby generating small-scale features in the balanced fields. We examine the detailed energy transfer pathways in wave-dominated flows and thereby develop a generalized small Rossby number geophysical turbulence phenomenology, with the two-mode (barotropic and one baroclinic mode) quasi-geostrophic turbulence phenomenology being a subset of it. The present work therefore shows that inertia-gravity waves would form an integral part of the geophysical turbulence phenomenology in regions where balanced flow is weaker than gravity waves.

Sunday, November 24, 2019 3:48PM - 5:32PM —
Session G40 Turbulent Boundary Layers: Curvature and Pressure Gradients I

3:48PM G40.00001 Direct numerical simulation of turbulent flows over curved walls with adverse pressure gradient

Abhiram Aithal, Antonino Ferrante, University of Washington, Seattle — Flow separation over curved walls with adverse pressure gradient (APG) occurs in many aerodynamic applications. However, the physical mechanisms of turbulence over curved bodies with APG are not yet well understood and the wall models employed in Reynolds-averaged Navier-Stokes (RANS) and large-eddy simulations (LES) for such flows need to be improved. In order to provide the necessary statistics for the validation of such models and explain the physical mechanisms of such flows, we have performed direct numerical simulations (DNS) over curved walls with APG. First, we have developed a fully-explicit and direct pressure-correction method to solve the incompressible NS equations in orthogonal curvilinear coordinates called FastRK3. FastRK3 is a three-stage, third-order Runge-Kutta projection-method which requires solving the Poisson equation for pressure only once per time step. Then, we have verified and validated FastRK3 with several test-cases and against available experiments. Last, we have performed a DNS study of the turbulent separated flow over a curved ramp and studied the dynamics of its turbulence kinetic energy.

4:01PM G40.00002 Non-equilibrium three-dimensional boundary layers at moderate Reynolds numbers

Marco G. Giometto, Department of Civil Engineering and Engineering Mechanics, Columbia University, New York 10027, Adryan Lozano-Duran, Center for Turbulence Research, Stanford University, California 94305, George I. Park, Department of Mechanical Engineering and Applied Mechanics, University of Pennsylvania, Pennsylvania 19104, Parviz Moin, Center for Turbulence Research, Stanford University, California 94305 — Non-equilibrium wall turbulence with mean-flow three-dimensionality is ubiquitous in geophysical and engineering flows. Under these conditions, turbulence may experience a counter-intuitive depletion of the turbulent stresses, which has important implications for modeling and control. Current turbulence theories are established mainly for statistically two-dimensional equilibrium flows and cannot predict such a behavior. In the present work, we propose a multi-scale model explaining the response of non-equilibrium wall-bounded turbulence under the imposition of three-dimensional strain. The analysis is performed via direct numerical simulation of turbulent channels at friction Reynolds numbers up to 1000. We show that scaling properties of the Reynolds stress are consistent with a model comprising momentum-carrying eddies with sizes and time scales proportional to their distance to the wall. We further demonstrate that the reduction in Reynolds stress follows a spatially and temporally self-similar evolution caused by the relative horizontal displacement between the core of the momentum-carrying eddies and the flow layer underneath.

This investigation is funded by the Office of Naval Research (ONR), Grant N00014-16-S-BA10
4:14PM G40.00003 Characteristics of the so-called uniform momentum zones and vortical fissures in a turbulent boundary layer downstream of an adverse pressure gradient, ALIREZA EBADI, CHRISTOPHER WHITE, University of New Hampshire — Particle image velocimetry (PIV) data acquired in a turbulent boundary layer downstream of an adverse pressure gradient at high Reynolds number is analyzed to study the characteristics of the so-called uniform momentum zones (UMZs) and vortical fissures (VFs) in the inertial region of the flow (where the viscous forces are subdominant). These characteristics, which include the wall-normal width of the UMZs and VFs, the velocity jump and vorticity across VFs, among others, are compared to those of the zero pressure gradient boundary layer at similar Reynolds number at the same facility.

4:27PM G40.00004 A new correlation for predicting transition Reynolds number of varying turbulence intensity with pressure gradient, WEI-TAO BII, MENG-JUAN XIAO, FAN TANG, ZHEN-SU SHE, Peking Univ. — We report a new finding on the correlation predicting the laminar-turbulent transition of a boundary layer for varying incoming turbulence intensity and pressure gradient. The new correlation displays a simple scaling of the transition Reynolds number on the incoming turbulence intensity, much simpler than previously proposed empirical ones, owing to an introduction of a transition central location parameter by our newly proposed symmetry-based description of the laminar-turbulent transition. Excellent agreement between the theory and the measurement/simulation data is found for the skin friction and wall heat flux distributions throughout the transition, as accurately validated by the T3-series flat-plate transition experiment and computation data. Owing to its simplicity, the transition model using the current correlation may have significant engineering interest. Furthermore, the results demonstrate that one may uncover simple similarity law governing the transition onset for both natural and bypass transitions, once relevant (statistical) multilayer structure of TBL is represented. The discovery of such similarity law does not require detailed analysis of complex instability mechanisms of the transition. Future work would be to quantify more transition effects, e.g., those in hypersonic engineering flows, which has been extremely difficult via the traditional approach.

1 NSF-CAREER 11377008, 11452002

4:40PM G40.00005 The effect of concave surface curvature on supersonic turbulent boundary layers, CHRISTIAN LAGARES, HPCVLab U. of Puerto Rico-Mayaguez, KENNETH JANSEN, U. of Colorado-Boulder, GUILLERMO ARAYA, HPCVLab U. of Puerto Rico-Mayaguez — Unsteady 3D turbulent boundary layers that evolve along the flow direction exhibit a streamwise non-homogeneous condition and pose enormous computational challenges. The reasons are as follows: (i) full spectrum resolution of turbulence, (ii) accurate time-dependent inflow turbulence information, and (iii) compressibility effects. Moreover, accounting for the effects of wall-curved driven pressure gradient adds significant complexity to the problem. In this presentation, we will show recent Direct Numerical Simulation (DNS) with high spatial/temporal resolution of supersonic spatially-developing turbulent boundary layers (SDTBL) subject to strong concave curvature ($\delta_{wall}/R \approx -0.083$, R is the curvature radius) and Mach $= 2.86$, which are of crucial importance in aerospace applications (e.g., unmanned high-speed vehicles and scramjets). The prescribed curved geometry is based on the experimental study by Donovan et al. (J. Fluid Mech., 259, 1-24, 1994). Turbulent inflow conditions are based on extracted data from a previous DNS over a flat plate (precursor). The extensive DNS information will shed important light on the transport phenomena inside turbulent boundary layers subject to strong deceleration or Adverse Pressure Gradient (APG) caused by concave walls.

1 NSF-CAREER 1847241, AFOSR FA9550-17-1-0051, XSEDE CTS170006

4:53PM G40.00006 ABSTRACT WITHDRAWN —

5:06PM G40.00007 Adaptive simulations and experiments of the turbulent flow around a NACA 4412 profile at high $Re^1$, FERMIN MALLOR, ALVARO TANARRO, EDA DOGAN, AGASTYA PARikh, NICOLAS OFFERMANS, ADAM PEPLINSKI, RAMIS ORLÜ, RICARDO VINUESA, PHILIPP SCHLATTER, Linne FLOW Centre, KTH Mechanics — Turbulent boundary layers (TLBs) under strong pressure gradients (PG), as appearing on wing surfaces, are still an active research topic. The NACA 4412 has been a benchmark airfoil in the study of PG-TLBS as its surface pressure distribution is only weakly dependent on Reynolds number ($Re$) at moderate angles of attack (AoA). This allows to decouple PG (history) and $Re$ effects affecting the development of the TLB. Using the high-order spectral-element method code Nek5000, large-eddy simulations of a NACA 4412 profile at a $5^\circ$ AoA are carried out. Adaptive Mesh Refinement (AMR) is used to generate a non-conformal mesh highly refined in the boundary layer and wake regions and avoiding the over-refinement of the far-field typical of structured conformal meshes. A larger domain is used while reducing by a factor of three the number of grid points, allowing to increase $Re$, aiming for 1,640,000 used in the experiments of the 1970s and 80s on the same airfoil. Such well-validated simulation data can then be used to understand better the effect of PG and surface curvature, and allow the calibration of turbulence and wall models. Moreover, an experimental campaign is planned in the MTL wind tunnel at KTH, for which wall-liners are designed aiming at reducing blockage effects.

1 Funding by the Knut and Alice Wallenberg Foundation and the ExaFLOW H2020 Project is acknowledged.

5:19PM G40.00008 Measurements in the near-wake of a turbulent wing-body junction, J. KLEWICKI, J.H. LEE, S. ZIMMERMAN, J. MONTY, University of Melbourne — Turbulent wing-body junction flows arise from the interaction between a turbulent boundary layer and a surface mounted streamlined obstacle, and are frequently encountered in aerodynamic and hydrodynamic applications. This study uses two-dimensional, three component stereo PIV measurements to investigate the statistical and instantaneous properties of a wing-body near-wake at moderate Reynolds number. The measurements were acquired in an open-return boundary layer wind tunnel at the University of Melbourne. The approach zero pressure gradient boundary layer at $f_0 = 10,000$ interacts with a Rood wing composed of a 3.2 elliptical leading edge joined to a NACA 0020 profile, with a chord length $C = 168$mm, maximum thickness $T = 40$mm, and height $H = 80$mm. The freestream velocity was about 20 m/s. Three component velocity measurements were collected at five different streamwise locations ranging from 0.075C to 1C downstream of the trailing edge of the airfoil, whose angle-of-attack was also varied from -15 to +15 degrees in increments of 5 degrees. Results focus on the three dimensional structure of the near-wake flow and the influence of the necklace vortices that form in front of the body and pass through the wake. Variations with angle of attack are described.

1 This research was support by the Office of Naval Research, grant no. N0014-17-1-2307

Sunday, November 24, 2019 3:48PM - 5:32PM — Session G41 CFD: General I
to large uncertainty. In ongoing work, we are performing high-fidelity computational fluid dynamics simulations to investigate reducing the ventilation patterns is represented using both single-sided and cross-ventilation models, while uncertainty in model parameters is accounted for sizes and locations of openings. A low-fidelity model is used within a second-order probability framework: uncertainty due to inconsistent validation a computational framework with uncertainty quantification to predict ventilation rates in a representative slum home with different pattern on both the configuration of the home, and on the highly variable operating conditions defined by weather and occupancy. In this study, Bangladesh, LAURA KWONG, Stanford University, FOSIUL NIZAME, International Centre for Diarrhoeal Disease Research, Bangladesh, STEPHEN LUBY, CATHERINE GORLE, Stanford University — Improved ventilation in slum housing could reduce the incidence of pneumonia, which is the leading cause of death in children under five. The goal of this project is to assess the effectiveness of different ventilation strategies for low-income housing in Dhaka, Bangladesh. One of the main challenges identified in field experiments is the dependency of the ventilation pattern on both the configuration of the home, and on the highly variable operating conditions defined by weather and occupancy. In this study, we validate a computational framework with uncertainty quantification to predict ventilation rates in a representative slum home with different sizes and locations of openings. A low-fidelity model is used within a second-order probability framework: uncertainty due to inconsistent ventilation patterns is represented using both single-sided and cross-ventilation models, while uncertainty in model parameters is accounted for using a second-order framework. The numerical model is validated using CFD results of NACA0012 airfoil from Sun.C et.al., 2015. The hydrodynamic forces are obtained by varying the angle of attack and side slip angles of the body from -20° to 20°, for flow velocity of 0.4m/s. The lift-to-drag ratios, flow physics around the wing are attacked and the trajectories are simulated (using in-house code) to arrive at an optimum L/D for increased endurance.

1Assistant Professor, Department of Ocean Engineering, Indian Institute of Technology Madras, India
2Ph.D. Scholar, Department of Ocean Engineering, Indian Institute of Technology Madras, India

4:14PM G41.00003 Modeling ventilation in an urban-slum home in Dhaka, Bangladesh1, YUNJAE HWANG, Stanford University, MAHAMUDUR HASAN, International Centre for Diarrhoeal Disease Research, Bangladesh, LAURA KWONG, Stanford University, FOSIUL NIZAME, International Centre for Diarrhoeal Disease Research, Bangladesh, STEPHEN LUBY, CATHERINE GORLE, Stanford University — Improved ventilation in slum housing could reduce the incidence of pneumonia, which is the leading cause of death in children under five. The goal of this project is to assess the effectiveness of different ventilation strategies for low-income housing in Dhaka, Bangladesh. One of the main challenges identified in field experiments is the dependency of the ventilation pattern on both the configuration of the home, and on the highly variable operating conditions defined by weather and occupancy. In this study, we validate a computational framework with uncertainty quantification to predict ventilation rates in a representative slum home with different sizes and locations of openings. A low-fidelity model is used within a second-order probability framework: uncertainty due to inconsistent ventilation patterns is represented using both single-sided and cross-ventilation models, while uncertainty in model parameters is accounted for using a second-order framework. The numerical model is validated using CFD results of NACA0012 airfoil from Sun.C et.al., 2015. The hydrodynamic forces are obtained by varying the angle of attack and side slip angles of the body from -20° to 20°, for flow velocity of 0.4m/s. The lift-to-drag ratios, flow physics around the wing are attacked and the trajectories are simulated (using in-house code) to arrive at an optimum L/D for increased endurance.

1This research is supported by an EVP Grant from the Stanford Woods Institute for the Environment.

4:27PM G41.00004 Effect of detaching force on water retention on a surface1, NEDA OJAGHLOU, DUSAN BRATKO, HOOMAN V. TAFRESHI, ALENKA LUZAR, Virginia Commonwealth University — The interaction of droplets with a surface is crucial in engineering applications such as coating, air filtration, and liquid transport in microfluidic devices. Despite recent advances in fluid mechanics and surface science, the mechanism of droplet detachment from the surface and the amount of residue left behind are not formulated well. When the droplets are forcibly removed from a surface, the ease of detachment strongly depends on the droplet volume and the rate of the removal. Experiments and continuum level calculations have so far been unable to resolve the time-dependent dynamics of droplet detachment and the role of the applied force as the key determinant of the volume of the droplet residue on the surface. We present a comprehensive study for predicting the force required to detach the water droplet from a graphene surface through the Molecular Dynamics (MD) simulations. Our results show that the minimum detaching force (per unit mass) decreases with the volume of the droplet and increases with the strength of water-surface adhesion. We also determined the amount of residue on the surface after detachment for different forces and water-carbon interactions. We observed that as the droplet size increases, a bigger residue remains on the surface. We found that the maximum amount of residue can be observed by applying the minimum force of detachment in contrast to experimental and MD results for droplet detachment from the curved surfaces where intermediate force was found to maximize the water retention.

1Supported by U.S. Department of Energy (No. DE-SC 0004406)

4:40PM G41.00005 A reduced order model for prediction of aerodynamic loads on an unmanned aerial system with hybrid quadcopter biplane configuration, MORTEZA HEYDARI, HAMID SADAT, University of North Texas — Army Research Lab (ARL) has recognized Unmanned Aerial Systems (UAS) with Vertical Take-Off and Landing (VTOL) to be capable of delivering paramount tasks such as intelligence, surveillance, electronic attack, etc. in the future. Advances in manufacturing and material technologies have opened a new design space for novel VTOL UAS configurations but significant knowledge gaps still exist to design a configuration which can achieve the desired performance. ARL designed research VTOL UAS platforms called Common Research Configuration (CRC) with hybrid quadcopter biplane concept to enable a comprehensive study on such aircraft. Inherent to all VTOL UAS such as CRC platforms is the need for a transition from hover to forward flight and forward flight back to hover throughout a mission. This transition produces highly non-linear loads on the wings due to the rotor-wing interactions and may present a significant challenge for robust control. The aim of this study is to develop a reduced order model (ROM) capable of predicting loads on CRC-3, the smallest size of the CRC generation with 3lb weight. A set of data obtained from hundreds of CFD simulations for a wide range of conditions are used as the training set for a neural network. The predicted loads by the developed ROM show good agreement with the test set. Additional data from the dynamic body equations are coupled with the ROM to investigate the CRC-3 flight dynamics. CFD simulations are conducted using our in-house solver, CFDFoam.
4:53PM G41.00006 Simulation of three dimensional aeronautical flow using the SED-SL algebraic turbulence model. 1, 2  TAN-TAN DU, MENG-JUAN XIAO, WEI-TAO BI, ZHEN-SU SHE, Peking Univ. — The SED-SL model specifies a multilayered stress length (SL) function in the wall-normal direction (which depicts the eddy viscosity), with slowly varying parameters along the streamwise direction simulating the entire turbulent boundary layer (TBL) including transition. After successful simulations of transitional flat-plate (APS meeting, 2016), airfoils (APS meeting, 2017), and 3-D aeronautical flows (APS meeting, 2018), here we report an in-depth study of the SED-SL model in simulating flows over M6 wing and DLR-F4 wing-body. For the M6 wing, intuitive consideration suggests a mildly varying buffer layer thickness in the spanwise direction, yielding a more accurate prediction of the shock locations and their merging above the wing surface, and thus a better prediction of the aerodynamic forces than those of the S-A and SST models. A comparison with a large eddy simulation of the flow shows also a superior prediction of the flow structures and Reynolds stress distribution. For the DLR-F4 wing-body, a significant improvement also is obtained with the new model compared to the SA and SST, in the prediction of the flow structures near the wing-body juncture and wing tip. These results demonstrate that the SED-SL model captures the right and simple (multilayer) structure of TBL, and is adaptable to aeronautical flows with not only high accuracy and efficiency for assisting design, but also more knowledge about the flow physics.

1NSFC Grants 11372008, 11452002

5:06PM G41.00007 Computational Investigations of Flow Past Axially Aligned Rotating Cylinders1, 2, 3 IGBAL MEHMEDAGIC, PASQUALE CARLUCCI, LIAM BUCKLEY, DONALD CARLUCCI, U. S. Army ARDEC, Picatinny Arsenal, NJ, SIVA THANGAM, Stevens Institute of Technology, Hoboken, NJ — Projectiles with free spinning segments are often used in smart munitions to provide effective control, stability and target guidance. Computational investigations are performed for flow past cylinders aligned along their axis where either the middle or base segment freely spins while attached to a non-spinning fore and/or aft body. The energy spectrum is modified to incorporate the effects of swirl and rotation using a parametric characterization of the model coefficients. An efficient finite-volume algorithm is used to solve the time-averaged equations of motion along with the modeled form of transport equations for the kinetic energy and the scalar form of turbulence dissipation. Experimental data for a range of spin rates and free stream flow conditions obtained from subsonic wind tunnel for flow past axially aligned cylinders with spinning segments are used to validate the computational findings.

1This work was funded in part by U. S. Army ARDEC

5:19PM G41.00008 Optimization of Pressurized Water Reactor Size and Coolant Flow Rate1, 2 SCOTT WAHLQUIST, Student, BRYAN LEWIS, Professor — In nuclear reactors, the heat generated in the system must be removed as fast as it is produced to operate in steady state. To accomplish this, liquid or gaseous coolant is moved through the core with pumps. In a typical pressurized water reactor (PWR), the standard inlet temperature is 290°C and the standard outlet temperature is 325°C. Using RANS modeling, a heat transfer CFD analysis was conducted on a basic structure of a PWR (neutron shield panel, nuclear core, and vessel wall) to show how water flows through the reactor and to determine the volumetric flow rate that creates the necessary convection to produce the inlet and outlet water temperature differential. The reactor was then scaled and the simulation was then repeated to observe the change in required flow rate. A plot comparing the size of the reactor to the required volumetric flow rate was made to determine the minimum and maximum reactors sizes with volumetric flow rates that don’t exceed the max allowable flow rate of a typical PWR pump and still produces the outlet temperature.

1BYU-I Undergraduate Student Research Fund

Sunday, November 24, 2019 3:48PM - 5:32PM
Session G42 Boundary Layers: Wind Turbine Interactions I 6e - Nicholas Hamilton, National Renewable Energy Laboratory

3:48PM G42.00001 Data Assimilation for the Prediction of Wake Trajectories Within Wind Farms1, 2 MAXIME LEJEUNE, Universite catholique de Louvain (UCLouvain), MARION COQUELET, UCLouvain - U Mons, NICOLAS COUDOU, UCLouvain - U Mons - VKI, MAUD MOENS, PHILIPPE CHATELAIN, Universite catholique de Louvain (UCLouvain) — Wind turbine wake physics is by nature unsteady and highly sensitive to the local wind characteristics. While Large Eddy Simulations (LES) allow to accurately capture the flow at the wind farm scale for a wide range of atmospheric conditions, they still come at a prohibitive computational cost when it comes to online control. Based on the Dynamic Wake Meandering model, the presented model couples a meandering model and a wake speed deficit model in order to estimate the velocity field downstream the wind turbine. Focus is laid on limiting the number of input parameters by building on recent advances in flow sensing to feed the wake model with estimated inflow conditions. The remaining parameters are fine-tuned through online data assimilation techniques, thus adapting to inflow conditions. The performances of the resulting wake model are assessed using data recovered from high fidelity LES simulations.

1Funding from the European Research Council under the European Unions Horizon 2020 research program. Computational resources made available on the Tier-1 supercomputer funded by the Walloon Region.

4:01PM G42.00002 Analytical model for yawed turbine asymmetric wakes. 1, 2, 3 MICHELE GUALA, St. Anthony Falls Laboratory, CEGE, University of Minnesota, BINGZHENG DOU, LIPING LEI, PAN ZENG, Department of Mechanical Engineering, Tsinghua University — Due to large scale atmospheric variability and slow nacelle adjustments, wind turbines are predominantly operating in yawed conditions. In addition, to minimize turbine-turbine interactions and maximize spatially averaged energy production in wind power plant arrays, the yaw angle becomes a control variable for operators to steer the wake of front row units away from downwind rotors. Both mechanisms are expected to be relevant also for Marine Hydrokinetic (MHI) Turbines, experiencing large scale variability due to migrating bedforms in fluvial or tidal flows. We present a new wake model able to predict the location of the maximum velocity deficit, and the asymmetric distribution of the mean velocity about the wake center, for different downwind distances (Dou et al., 2019). The model has been calibrated using wind tunnel and open channel flow experiments with yawed miniature turbines of different shape, size and operating conditions. The key advantages of the proposed wake model are: i) the model requires only the thrust coefficient in un-yawed conditions, ii) the required parameter describing the wake expansion has been observed to be weakly depending on specific incoming flow condition, iii) the parameter describing the wake asymmetry can be estimated based on the yaw angle and the thrust coefficient. Thus, input parameters of the proposed model are for the most part limited to the turbine geometry, its operating conditions, and the associated thrust coefficient.
A simplified DWM model are compared with the LES results. To evaluate Taylor's frozen-flow hypothesis using the LES data of turbine wakes. The space-time correlations are examined. The predictions from our knowledge, the validity of Taylor's frozen-flow hypothesis for modeling wake meandering has not been fully evaluated. In this work, we have developed a meandering (DWM) model has been developed in the literature (Risø National Laboratory Technical Report, 2007, Risø-R-1607). In the DWM simulation (LES), which is able to predict the energetic coherent structures of turbine wakes, on the other hand, cannot be directly applied to the design of wind farms because of its high computational cost. To fast predict the meandering motion of turbine wakes, the dynamic wake meandering (DWM) model has been developed in the literature (Risø National Laboratory Technical Report, 2007, Risø-R-1607). In the DWM model, the meandering motion is taken into account by modeling wakes as passive scalars based on Taylor's frozen-flow hypothesis. To the best of our knowledge, the validity of Taylor's frozen-flow hypothesis for modeling wake meandering has not been fully evaluated. In this work, we evaluate Taylor's frozen-flow hypothesis using the LES data of turbine wakes. The space-time correlations are examined. The predictions from a simplified DWM model are compared with the LES results.

Numerical Perspectives on Wind Turbine Wakes

Assessment of wake superposition models through wind tunnel tests and LiDAR measurements.

Impact of utility-scale wind turbine wakes on surface fluxes

Impact of utility-scale wind turbine wakes on surface fluxes

Impact of utility-scale wind turbine wakes on surface fluxes

UAS Swarming for Three-Dimensional Wake Measurements of Building and Turbine Wakes
8:00AM H01.00001 Dominant Resonance in Parametric Subharmonic Instability of Internal Waves

REZA ALAM, YONG LIANG, LOUIS-ALEXANDRE COUSTON, QIUCHEN GUO, UC Berkeley — Parametric Subharmonic Instability (PSI) is one of the most important mechanisms that transfer energy from tidally-generated long internal waves to short steep waves. Breaking of short waves results in diapycnal mixing through which oceanic abyssal stratification is maintained. It has long been believed that PSI is strongest between a primary internal wave and perturbative waves of half the frequency of the primary wave. Here, we show that this is not the case. Specifically, we show that neither the initial growth rate nor the maximum long-term amplification occur at the half frequency, and demonstrate that the dominant subharmonic waves have much longer wavelengths than previously thought.

8:13AM H01.00002 Resonance and transmission of axisymmetric internal wave modes

PHILIPPE ODIER, SAMUEL BOURY, Laboratoire de Physique, Ecole Normale Superieure de Lyon, France

THOMAS PEACOCK, Department of Mechanical Engineering, MIT, USA — To date, axisymmetric internal wave fields, relevant to atmospheric waves generated by storm cells and oceanic near-inertial waves produced by surface perturbations, have been experimentally produced using an oscillating sphere or torus as the source. Here, we use a wave generator configuration capable of exciting axisymmetric internal wave modes of arbitrary radial form. The efficiency of the wave generator is measured through careful estimation of the wave amplitude based upon group velocity arguments, and the effect of vertical confinement is considered to induce resonance, identifying a series of experimental resonant peaks agreeing well with theoretical predictions. In the vicinity of resonance, the waves undergo a transition to nonlinear behavior. In a second step, we investigate transmission of these modes in nonlinear stratifications. Two configurations are studied: in the case of a free incident wave, a transmission maximum is found in the vicinity of evanescent frequencies. In the case of a confined incident wave, resonant effects lead to enhanced transmission rates from upper to lower layer. We consider the oceanographic relevance of these results by applying them to an example oceanic stratification from the Arctic, finding that there can be real-world implications.

1We acknowledge support by the Agence Nationale de la Recherche through grant "DisET" No. ANR-17-CE30-0003

2APS Membership pending

8:26AM H01.00003 Numerical Simulations of the Internal Waves Produced by a Submerged Body in a Stratified Fluid

LAURA BRANDT, DEVIN CONROY, JAMES ROTTMAN, Leidos — We attempt to gain some insight into the modeling of internal waves produced by a submerged body traveling horizontally at high Reynolds number in a strongly stratified fluid by comparing numerical simulations with linear theory. Two types of internal waves are generated by the horizontal motion of a body in a stratified fluid: lee waves, which are steady in a reference frame moving with the body and are generated by the motion of the body itself, and wake waves, which are unsteady and generated by the turbulent wake. Traditionally the lee waves have been represented in linear theories by a source singularity in the continuity equation, but recently it has been argued that a body force needs to be added to the momentum equations in order to accurately represent the lee waves. We test this latter hypothesis directly by comparing linear theory with our numerical simulations in which either a free-slip or no-slip boundary condition at the surface of the body is imposed. The free-slip boundary condition represents a body with no downstream recirculation region, so the source singularity should be accurate, and the no-slip boundary condition would have a downstream recirculation region, so that the body force should be necessary.

8:39AM H01.00004 Linear and nonlinear fate of an axisymmetric inertial wave attractor

SAMUEL BOURY, THIERRY DAUXOIS, SYLVAIN JOUBAUD, PHILIPPE ODIER, Univ Lyon, ENS de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France

EVENY OGREN, University of Gothenburg, Sweden

KUNIHIRO SUGIURA, University of Tokyo, Japan

Iliyas Sibgatullin, Lomonosov Moscow State University, Moscow, Russia — For a few decades now, numerous studies have been devoted to the properties of inertia-gravity wave reflection. Since the angle of propagation of these waves is set by the ratio of their frequency to the buoyancy or rotation frequency, the reflection on a wall does not follow the usual Snell-Desartes law. In particular, in a confined trapezoidal domain, the wave beam experiences a focusing effect and eventually ends on a limit trajectory called attractor. Experimental and numerical studies have shown evidence of this phenomenon for internal waves in 2D geometry. Due to the local energy focusing, nonlinear triadic cascades occur in the branches of the attractor, leading to energy transfer between scales. More recently, geometric and 3D aspects of internal wave attractors have been explored using Direct Numerical Simulations of inertial waves. The DNS pictured an axisymmetric inertial wave attractor, in a trapezoidal cylindrical domain, with focusing and defocusing effects caused by wave reflections and by the radial geometry itself. Wave instability occurs while forcing the attractor and leads to an azimuthal symmetry breakdown. Using an apparatus relevant for axisymmetric wave generation, we produce an inertial wave attractor in a cylindrical domain and we explore its properties.

1Supported by ANR Grant "DisET" No. ANR-17-CE30-0003.

8:52AM H01.00005 Modal decomposition of polychromatic internal wave fields in arbitrary stratifications

MORRIS FLYNN, University of Alberta, ALEXIS KAMINSKI, Applied Physics Laboratory, University of Washington — Internal waves e.g. those produced by tidal sloshing over bathymetry play a crucial role in the energetics of the oceanic overturning circulation. Understanding their spatial and temporal structure, which depend on both the details of the forcing topography and the forcing frequency, is essential in predicting where mixing may occur, details of which remain poorly understood. Past work has largely focused on the case of a monochromatic wave-field; however, tides are composed of multiple frequency constituents. Here we present an approach by which the modal structure of a polychromatic internal wave-field may be computed from velocity data without any a-priori knowledge of the details of the forcing topography. We consider wave-fields in both uniform and vertically-varying stratification, and show using synthetic data that our approach is able to accurately reconstruct the vertical mode strengths. The sensitivity of our approach to noise and vertical resolution is also examined.

1NSERC
The instabilities of spatially monochromatic internal waves have been well-studied in the past decades, owing to their fundamental nature and relevance to geophysical processes. On the other hand, internal wave beams with locally confined spatial profile have only recently gained appreciation for their distinct properties and instability mechanisms. Such wave beams provide a more realistic setting for the study of internal wave instabilities in the ocean as they naturally arise, for instance, from the interaction of the barotropic tide with bottom topography. Owing to their additional complexity, internal wave beam instabilities have thus far been analyzed primarily in weakly nonlinear contexts, with the most famous being the parametric subharmonic instability. Here, we instead investigate the stability of finite-amplitude internal wave beams to two-dimensional perturbations using Floquet theory. We then compare these theoretical results with novel experimental observations.

1Supported by NSF Graduate Research Fellowship grant 1122374 and grant DMS-1512925.

Experimental results on the long-time spatial and temporal development of Triadic Resonance Instability

Various mechanisms have been proposed to explain how internal waves transfer energy across the wavenumber and frequency spectrum and eventually contribute to small-scale turbulence and mixing. Triadic Resonance Instability (TRI) has become increasingly recognized as one of these mechanisms. This talk will focus on new experimental investigation into the temporal and spatial evolution of this instability. Experiments have been conducted using a wave tank set-up, featuring a flexible horizontal boundary condition which is driven by an array of independently controlled actuators. This allows for varying amplitude, frequency and wave-length in both the spatial and temporal domain. We present results utilizing this ability to vary the forcing parameters during the course of an experiment and investigate what effects this has on the evolution of TRI.

1National Environmental Research Council (NERC)

Large internal waves in deep water: models, numerics and experiments

We investigate the propagation of large solitary internal waves in continuously salt-stratified water close to homogeneous two-layer configurations. When the lower, denser layer is much deeper than the one at the top. Experiments are performed in the UNC wave tank, with data collected via PIV and LIF, from both fixed locations and by a cart moving with the waves. Predictions from an optimized two-layer fully nonlinear model are compared with the experiments and direct numerical simulations of Euler equations in two dimensions.

1DMS-1910824, DMS-1517879, ONR N00014-18-1-2490

Joint effects of topography and rotation in internal solitary waves

An asymptotic, adiabatic theory for the evolution of an internal solitary wave governed by the variable-coefficient, rotation-modified Gardner equation (Korteweg-de Vries with cubic nonlinearity) is developed and used to explore the joint effects of variable topography and rotation on the evolution of this instability. Experiments have been conducted using a wave maker, featuring an array of independently controlled actuators. We then compare these theoretical results with novel experimental observations.

1Supported by the Office of Naval Research grant N00014-18-1-2542.

Examining the initial development of convective instability in a three-dimensional shoaling internal solitary wave of depression in over gentle slopes

A convectively unstable internal solitary wave (ISW) of depression, shoaling over gentle slopes (<3%), is examined through fully nonlinear and non-hydrostatic simulations. These simulations are based on a high resolution/accuracy deformed spectral multidomain penalty method flow solver. The convective instability occurs due to a sudden increase in the propagation speed, below the maximum horizontal wave induced velocity, the wave retains its nearly symmetric shape as it shoals. Subsequently, an unstable region develops characterized by the entrainment of heavier-over-light fluid, in the form of a recirculating, or trapped, core. The preceding convective instability is attributed to the stretching of the near-surface vorticity layer of the baroclinic background current from the propagating ISW. According to recent field observations in the South China Sea, this region may persist for more than 10 km and drive turbulent-induced mixing, estimated to be up to four times larger than that in the open ocean. Motivated by such observations, emphasis in this presentation is placed on the onset of the 3D convective instability as the ISW shoals. The ISW propagates in the normal-to-isobath direction. The initial 3D instability is visualized via a transitional structure that develops in the lateral direction. The evolution of the lateral instability is compared with the convective overturn of the core. As such, a preliminary understanding of the formation of recirculating cores in ISWs is obtained.

1NSF OCE 1634257
10:10AM H01.00011 Propagation and Breaking of Three-Dimensional Boussinesq Wave Packets without and with Rotation

1, ALAIN GERVAIS, Department of Mathematical and Statistical Sciences, University of Alberta, Canada, QUINLAN EDE, Department of Earth and Atmospheric Sciences, University of Alberta, Canada, GORDON SWATTERS, Department of Mathematical and Statistical Sciences, University of Alberta, Canada, TON VAN DEN BREMER, Department of Engineering Science, University of Oxford, United Kingdom, BRUCE SUTHERLAND, Departments of Physics and of Earth and Atmospheric Sciences, University of Alberta, Canada — Internal gravity waves (IGWs) propagate vertically and horizontally within stably stratified fluids. As IGWs propagate vertically, nonlinear processes lead to instabilities that may cause a wave to overturn and eventually break, thus irreversibly depositing momentum to the background flow. Even before breaking, moderately large amplitude IGWs induce a mean flow that interacts nonlinearily with the waves, Doppler-shifting their frequency and altering the height at which the waves would have otherwise overturned. Here we derive explicit formulae for the induced flows of localised wavepackets influenced by Coriolis forces. Numerical simulations are initialised with quasi-monochromatic wave packets with the predicted induced flow superimposed. Simulations with small amplitude wave packets confirm that the prediction captures the induced flow. In simulations with larger amplitude waves, the nonlinear interactions between the waves and the flows they induce result in much lower overturning heights than predicted by linear theory.

1Natural Sciences and Engineering Research Council of Canada (NSERC)

10:23AM H01.00012 Wave focusing and related multiple dispersion transitions in plane Poiseuille flows

8:00AM H02.00001 On the mechanism that sustains intermittent attached cavitation inception in a boundary layer without flow separation.

Monday, November 25, 2019 8:00AM - 10:36AM — Session H02 Multiphase Flows: Cavitation and Aerated Flows

8:13AM H02.00002 Role of Compressibility on the Shedding Dynamics of a Cavitating Wake

8:26AM H02.00003 Computational Modeling of Air Cavities under Solid Bodies in Water Flow

1This material is based upon research supported in part by the National Science Foundation under Grant No. 1800135 and in part by the U. S. Office of Naval Research under Award No. N00014-17-1-2553.
8:39AM H02.00004 Cavitation dynamics in the wake of a backward facing step¹. ANUBHAV BHATT, HARISH GANESH, STEVEN L. CECCIO, University of Michigan, Ann Arbor, MI-48109 — The flow over a backward facing step is often used as a benchmark for experimental studies on separation and reattachment in hydrodynamic applications. The flow has a separating and reattaching shear layer forming a recirculating region. These shear layers are susceptible to cavitation, and can experience different dynamics depending upon the extent of the cavitation formed in the shear layer and contained within the recirculation bubble. Beginning with cavitation inception, a reduction in cavitation number results in different cavitation regimes such as fully developed cavitation, self-sustained cloud shedding and super-cavitation. This study focuses on quantifying the cavitation dynamics of the developed cavitation, at three Reynolds numbers (8.5*1E4, 10.6*1E4 and 12.7*1E4) using time-resolved X-Ray densitometry and high speed videography. In addition, static and unsteady pressure measurements are performed to understand the change in dynamics within the region of flow separation. The effect of compressibility of the vapor-liquid mixture is assessed by estimating the speed of sound based on the static pressure and void-fraction measurements at different cavitation conditions.

¹This work was supported by Office of Naval Research (ONR), MURI Grant N00014-17-1-2676, under program manager Dr. Ki-Han Kim.

8:52AM H02.00005 Investigation of turbulent cavitating flows in a small Venturi by fast X-ray imaging. GUANGJIAN ZHANG, Arts et Metiers ParisTech, MINMING GE, Virginia Tech, ILYASS KHLIFA, Arts et Metiers ParisTech, OLIVIER COUTIER-DELGOSHA, Virginia Tech — The cavitating flows created in a small Venturi nozzle are investigated based on ultra-fast x-ray imaging. The instantaneous velocities of the liquid and vapor are measured simultaneously by tracking seeding particles and vapor structures respectively while the vapor volume fraction is derived from the different x-ray attenuation. Wavelet decomposition with appropriate thresholds is used to separate seeding particles from vapor structures, so that image cross-correlations could be applied on the two phases separately. This study presents data on mean velocity and void ratio field, statistical turbulent quantities in three different cavitation levels with the same reference velocity. A type of cavitation associated with a weak but persistent re-entrant jet is described. The comparison between the cavitation and the non cavitation flow field shows that the averaged flow field is significantly altered by the presence of cavitation and the vapor formation near the throat area is observed to suppress velocity fluctuations.

9:05AM H02.00006 Thermodynamic effects on Venturi cavitation characteristics¹. ZHIGANG ZUO, HAOCHEN ZHANG, Tsinghua Univ., KNUD AAGE MRCH, Technical Univ. of Denmark, SHUHONG LIU, Tsinghua Univ. — In studies using cold water as the working liquid, the thermodynamic effect of cavitation is usually ignored. However, in cryogenic liquids, refrigerants and high temperature water the thermodynamic effect is significant, and it suppresses the development of cavitation by reducing the temperature at the boundary of expanding cavitation bubbles. In this paper the thermodynamic effect is systematically studied by Venturi cavitation in a blow-down type tunnel for the first time, using water at temperatures up to relatively high levels, and at controlled dissolved gas content in the supply reservoir (measured by dissolved oxygen, DO). The cavitation characteristics are analyzed from the experiments, and the mean cavitation length is chosen as the quantity suited to reveal the thermodynamic effect. With an increase of the thermodynamic parameter, a decrease of cavitation length is observed, which is consistent with suppression of cavitation by the thermodynamic effect. Within the range of DO content tested, the DO content has little influence on the mean cavitation length and the unsteady cavitation characteristics, which is in contrast to the effect of gas content on cavitation nuclei generally.

¹The work was supported by National Natural Science Foundation of China (No. 51479083), National Basic Research Program of China (No. 613321), State Key Laboratory of Hydro Science and Engineering (Research Fund Programs, No: 2017-KY-03 and 2019-KY-04), and the Otto Muensted Foundation (Grant 15-81-1166).

9:18AM H02.00007 Thermal effects in cavitating flows. MARTIN PETKOVSEK, University of Ljubljana, DREW JACOBS, MINMING GE, OLIVIER COUTIER-DELGOSHA, Virginia Tech — The effects of temperature on hydrodynamic cavitation in water is investigated. Temperature is varied between 20C and 85C in a cavitating flow generated in a small-scale venturi type section of 1 mm characteristic dimension. The effects of the temperature on the large-scale periodical cloud shedding is investigated, as well as the evolution of the flow structure and dynamics. The measurements are based on high speed visualizations and 2D2C PIV. A competition between two different effects, namely the Reynolds number change and the cavitation delay due to the so-called thermal effect, is observed. A critical temperature in the range 50C-60C is found and especially investigated.

9:31AM H02.00008 Control of cavity bubble in water entry using laser-induced cavitation. KYUSEONG CHOI, NAYOUNG KIM, GUWON SEON, WONTAE HWANG, HYUNGMIN PARK, Seoul National University — We investigate how the cavity attached to the metallic sphere in water entry changes when laser is irradiated. The sphere (radius, R=1, 2mm) is roughened (0.1-1µm in size) to generate a cavity even at a relatively low impact speed (Uo=1.5-3.3m/s). By varying the height of dropping position and irradiation time, that is speed and surface temperature (To=110-350°C) at the impact instant, we measure the cavity dynamics with a high-speed camera (the water is at room temperature). In the case of a shallow seal (R=1mm, Uo=1.5m/s), we classify two regimes of cavity growth (To=170-240°C) and destruction (To=240°C). In the destruction regime, microbubble emission boiling happens, so the cavity is destructed to numerous microbubbles. In the case of a deep seal (R=1mm, Uo=3.3m/s), the slight cavity growth occurs at To=130-150°C and considerable destruction of cavity bubble at To=170°C. At a transient of To=150-170°C, the deep seal changes to shallow seal with a slight destruction of cavity. As a change of cavity dynamics, the forces acting on the sphere is varied, which is estimated from measured cavity geometry and sphere trajectory. Finally, we suggest a mechanism of cavity growth and destruction according to Uo and To.

¹Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

9:44AM H02.00009 Air entrainment mechanisms of a forced plunging jet¹. SOPHIA RELPH, KEN KIGER, University of Maryland, AKASH DHURV, ELIAS BALARAS, George Washington University — Plunging jets play a major role in the quality of cast metal parts, as the pouring process can entrain metal oxides in much the same way as pouring water captures bubbles. These air and oxide pockets interfere with the metals crystal structure and can compromise strength and fatigue life. The mitigation of such defects is of great interest to foundries. Most research on plunging jets relevant to metal casting considers either smooth or passively disturbed jets that result from a turbulent nozzle state, with little characterization beyond the variance of the velocity. We know that at higher velocities, defects is of great interest to foundries. Most research on plunging jets relevant to metal casting considers either smooth or passively disturbed jets that result from a turbulent nozzle state, with little characterization beyond the variance of the velocity. We know that at higher velocities, jet disturbances play a large role in air entrainment, but the literature is inconclusive on the mechanism by which this occurs. The current work examines the role of carefully controlled forced disturbances on both a plunging jet and the pool surface, allowing us to correlate surface disturbance properties with air entrainment behavior. The current effort is focused on determining the size (wavelength and amplitude) of the disturbances, as well as the relative phaseing, on controlling the inception of air entrainment and the volume of the resulting air entrainment events. Results from a laboratory-scale air/water experiment and corresponding DNS will be presented.

¹Work sponsored by the DLA Troop Support Philadelphia, PA and the Defense Logistics Agency Fort Belvoir, VA under contract SP4701-18-D-1200
Homogeneous Equilibrium Mixture (HEM) Cavitation Model. JOSHUA BRINKERHOF, SAEED RAHBARIMANESH, University of British Columbia; IOANNIS KARATHANASSIS, MANOLIS GAVAISES, City, University of London — The commonly-used homogeneous equilibrium mixture (HEM) cavitation models lack accurate treatment of two phase flow properties at mixture saturation states, which leads to miscalculation of pressure and density in computational fluid dynamics (CFD) simulations. This issue can be addressed by employing a reliable barotropic equation of state (EOS) in the cavitation model. The current study presents a piecewise EOS that can accurately capture flow properties at liquid, vapor, and transition states. Implementing the proposed EOS in a HEM solver to simulate the cavitating flow of diesel in a throttle shows a better match against mass flow rates measured in experiments than a single-step EOS approach. The improvement is due to the enhanced functionality of the solver in relaxing sharp pressure and density gradients during condensation and vaporization processes. The lack of such capability in the single-step model seems to cause additional numerical diffusion in shear layers and interface regions, particularly in areas with stronger condensation, resulting in poorly predicted cavitation. The work is a step towards the main objective of improving HEM cavitation modeling in CFD of cryogenic fluids.

10:10AM H02.00011 Numerical study of cavitating flows around an immersed solid body.1. GIHUN SON, SEONGJIN HONG, Sogang University — A numerical approach is presented for cavitating flows around an immersed solid body. The single-fluid model for cavitating flows, which is based on a barotropic relation between fluid density and pressure variations in the liquid-vapor phase-change region, is coupled to a fictitious-domain (FD) method, where the immersed solid region is assumed to be filled with the surrounding fluid with a high viscosity. The FD method can be efficiently applied to complex body geometries without generating body-fitted meshes and does not need an explicit calculation of forces and torques acting on the solid boundary unlike immersed boundary methods. The conservation equations of mass and momentum with the compressibility effect in the cavitation region are solved by employing a projection method and a semi-implicit pressure correction method to avoid the serious time step restriction in weakly compressible flows. The present numerical method combined with a non-equilibrium k-ε turbulence model is tested through computations of cavitating flows around a hemispherical body and a wedge-shaped body, whose numerical results or experimental data are available in the literature. The numerical method is extended for cavitating flows around a moving solid body.

1 This work was conducted with the support of the Korea Environment Industry & Technology Institute (KEITI) through its Ecological Imitation-based Environmental Pollution Management Technology Development Project, and funded by the Korea Ministry of Environment (MOE) (2019002790006).

10:23AM H02.00012 Gelatin cavity dynamics in the wake of high-speed solid sphere impact.1. AKIHITO KIYAMA, Tokyo University of Agriculture and Technology, MOHAMMAD MANSOOR, NATHAN SPEIRS, Utah State University, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology, TADD TRUSCOTT, Utah State University, TUAT COLLABORATION, USU COLLABORATION — We investigate the impact and penetration of a small solid sphere onto gelatin at speeds greater than 100 m/s. High-speed videography allows us to capture the cavity dynamics in the wake of the sphere. Varying the gelatin concentrations affects the elastic response of the medium. The high-speed videography reveals several unique features of the cavity dynamics in gelatin when compared to water (e.g., the appearance of the texture on the wall of gelatin cavity, the attenuation of the vertical jet upon the pinch-off of gelatin cavity). We present a phase diagram that classifies the cavity pinch-off type using the elastic Froude (inertia vs. elasticity) and the elastic Grashof (gravity vs. elasticity) numbers (Akers & Belmonte, J. Non-Newtonian Fluid Mech., 2006), similar to the Weber and Bond numbers used for water entry (Aristoff & Bush, J. Fluid Mech, 2009). We also discuss the detailed dynamics of each cavity type based on high-speed images.

1 JSPS KAKENHI Grant number 16H08521 and 17H01246, and support from Tokyo University of Agriculture and Technology

Monday, November 25, 2019 8:00AM - 10:36AM
Session H03 Focus Session: Direct Numerical Simulations of Fluid Interfaces, Deformation and Break-Up in Turbulence 201 - Olivier Desjardins

8:00AM H03.00001 DNS of turbulent flows laden with deformable bubbles or droplets: Overview of methods. SAID ELGHOBASSI, University of California, Irvine — Turbulent flows laden with liquid droplets or gas/vapor bubbles are ubiquitous in nature and engineering applications. In nature, examples include rain, air bubbles in the upper ocean, and vapor bubbles in geysers. Engineering applications include liquid fuel sprays in all types of combustion engines, paint sprays, spray drying in the pharmaceutical industry as well as food processing, and water vapor bubbles in nuclear reactor cooling systems or those created by cavitation in the wakes of ship propellers, just to list a few. This lecture focuses on direct numerical simulations (DNS) of turbulent flows laden with droplets or bubbles. DNS of these flows are more challenging than those of flows laden with solid particles due to the surface deformation in the former. The numerical methods to be discussed are classified by whether the initial diameter, d, of the bubble/droplet is smaller or larger than the Kolmogorov length scale, η. The lecture discusses DNS of deformable small spherical bubbles/droplets (d < η) via a phenomenological model.[1]


8:13AM H03.00002 DNS of turbulent flows laden with deformable bubbles or droplets: Recent advances. SAID ELGHOBASSI, University of California, Irvine — Turbulent flows laden with liquid droplets or gas/vapor bubbles are ubiquitous in nature and engineering applications. In nature, examples include rain, air bubbles in the upper ocean, and vapor bubbles in geysers. Engineering applications include liquid fuel sprays in all types of combustion engines, paint sprays, spray drying in the pharmaceutical industry as well as food processing, and water vapor bubbles in nuclear reactor cooling systems or those created by cavitation in the wakes of ship propellers, just to list a few. We discuss recent advances in the numerical methods[1] of deformable large spherical bubbles/droplets whose initial sizes are larger than the Kolmogorov length scale, η. The methods include the Conservative Level Set (CLS), Volume of Fluid (VOF), Front Tracking (FT), Phase Field Model (PFM), and Lattice Boltzmann (LB).

8:52AM H03.00005 Direct numerical simulation of bubble-induced turbulence at high Reynolds number, ALICE JACCOD, Sorbonne Universite, ALESSIO INNOCENTI, Istituto Nazionale di Geofisica e Vulcanologia, STEPHANE POPINET, SERGIO CHIBBARO, Sorbonne Universite — Among the various kind of multiphase flows, bubbly flows represent a challenging and key field of investigation for their particular dynamics and their applications in several fields. Important experiments have been carried out in the last decades but a precise understanding of bubble-induced turbulence is still lacking. In the present work, a study of this phenomenon is presented, by performing fully resolved two and three dimensional numerical simulations of bubbles, rising up under the effect of buoyancy. Bubbles, initially placed at rest at the bottom of a channel, experience a large transfer rate with the liquid, resulting in an agitated turbulent motion, called pseudo-turbulence. Varying the physical parameters of the problem as the bubble volume fraction and increasing the Reynolds number, it's possible to outline a large phenomenology of the dispersed phase flow. An investigation of energy spectra and velocity fluctuations probability density function has been done, for both two and three dimensional cases. Moreover, we performed a scale-by-scale analysis of energy transfer, to identify the spatial scale within which a direct or inverse energy cascade is present.

9:18AM H03.00007 Breakup at the resolution limit, MARCUS HERRMANN, Arizona State University — The process of atomization is characterized by a vast range of time and length scales present in the flow. In fact, when the topological change of the phase interface occurs, i.e. a liquid structure breaks into smaller structures, the length scale goes to zero. Thus simulations of atomization are not able to resolve all length and time scales at all times. Typically, simulations are under-resolved during the final stages of breakup, relying on the properties of the numerical methods used, i.e., the properties of the methods’ inherent numerical errors, to capture the topology change events correctly. This reliance on numerical errors to reproduce physical processes is questionable, but unavoidable without dedicated, breakup models that do not rely on the local mesh resolution to initiate breakup. In this talk, we will discuss how different interface capturing techniques perform during the final stages of breakup, using the breakup of a ligament in a test bed as an example. The results can give insights into the drop size distributions obtained in detailed simulations of atomizers, especially concerning smaller drop sizes near the mesh resolution limit.

9:31AM H03.00008 Lattice Boltzmann Simulations of Interface Deformation and Breakup in Turbulent Flow Over Superhydrophobic and Liquid-Infused Surfaces1, AMIR-REZA RASTEGARI, RAYHANEH AKHAVAN, The University of Michigan — Interface deformation and breakdown in turbulent flow over Super-Hydrophobic (SH) and Liquid-Infused (LI) surfaces is investigated by Direct Numerical Simulation (DNS) using a two-phase, single relaxation time, free-energy lattice Boltzmann method. In this method, the dynamics of a diffuse interface is incorporated into the governing equations using a Peng-Robinson free-energy functional. This obviates the need for explicit tracking of the interface or pinning of the contact line. DNS studies were performed in turbulent channel flows with longitudinal microgrooves of width $15 \leq g^{1/10} \leq 64$ in base flow units, at solid fractions of $\phi_s = 1/16$ or 1/2 on both walls. Simulations were performed at a base flow friction Reynolds number of $Re_T \approx 222$, with viscosity ratios of $\mu_{ext}/\mu_{int} = 10$, 20 and 55, and Weber numbers of $10^{-3} \leq We_r \leq \mu_{ext}/\sigma \leq 10^{-2}$. Analysis of the results shows that interface deformation and contact line motion can significantly reduce the magnitude of drag reduction compared to DNS results obtained in turbulent flow with 'idealized', flat SH or LI interfaces. In addition, the simulations identify the conditions for contact line depinning and interface breakdown.

1Supported by M. R. Prince Foundation and NSF XSEDE Allocation TG-CTS070067N.
The non-zero inlet angle is shown to impact small-scale interfacial wave formation and breakup. Wavelet analysis is performed to the macro-scale jet dynamics, such as jet deflection and asymmetric deformation of the head of liquid jet. The simulations results of the global through high-fidelity simulation. The non-zero inlet angle induces an azimuthal variation of velocity in the liquid jet, which in turn influences many fuel injection applications, the liquid flow direction at the nozzle inlet is not perpendicular to the inlet plane. The effect of the angle is generally considered to be aligned with the nozzle axis. As a result, the near-field instability development is axisymmetric. Nevertheless, in many fuel injection applications, the liquid flow direction at the nozzle inlet is not perpendicular to the inlet plane. The effect of the angle between the inlet velocity and nozzle axis on the primary breakup of a round liquid jet is unclear and will be investigated in the present study through high-fidelity simulation. The non-zero inlet angle induces an azimuthal variation of velocity in the liquid jet, which in turn influences the macro-scale jet dynamics, such as jet deflection and asymmetric deformation of the head of liquid jet. The simulations results of the global features, including jet deflection angle and temporal evolution of penetration length, are compared with experimental data and good agreement is achieved. The non-zero inlet angle is shown to impact small-scale interfacial wave formation and breakup. Wavelet analysis is performed to characterize the azimuthal variation of the interfacial waves in the near field.

1 Supported by DoE/CASL.

10:10AM H03.00011 Characterizing the dynamics of complex multiphase flows undergoing topology changes. GRETAR TRYGGVASON, JIACAI LU, Johns Hopkins University — Although disperse multiphase flows, where one fluid appears as discrete drops or bubbles dispersed in another contiguous phase, have been widely studied, such flows are generally only seen when the volume fractions of the fluids are very different. For comparable void fractions, particularly if the fluid is turbulent, we expect to see a complex dynamic interface whose topology changes repeatedly as fluid masses coalesce and break apart. Describing and modeling such churn-turbulence flows is challenging. Approaches drawn from studies of heterogeneous solids, rheology, and premixed combustion provide some guidance, but do not cover the full complexities of a dynamic interface and fluid turbulence. The challenges involve both finding the appropriate statistical descriptions as well as the generation of coarser models. We discuss these challenges; review the various ways the flow can be described and modeled; and show examples from our recent work on complex turbulent buoyant two fluid flows in vertical channels and the breakup of periodic liquid jets.

1 Supported by DoE/CASL.

10:23AM H03.00012 Simulating high void fraction flows undergoing massive topology changes in vertical channels. JIACAI LU, GRETAR TRYGGVASON, Johns Hopkins University — Turbulent multiphase flows in vertical channels, where the topology of the interface between the different fluids repeatedly changes due to the breakup and coalescence of fluid masses, are examined by numerical simulations, using a finite volume/front-tracking method where the interface is tracked by connected marker particles and the flow equations solved on a regular structured grid. When a film of one fluid, separating blobs of a different fluid, becomes sufficiently thin, it is ruptured. At low volume fraction of one phase, one phase usually consists of disperse drops or bubbles, but as its volume fraction increases the interface structures changes from to more complex irregular shapes, where each fluid is often highly interconnected. Here the focus is on flows where the volume fractions are comparable. The evolution of various integral quantities, such as the average flow rate, wall-shear, and interface area and structure are monitored and compared for different governing parameters such as void fraction and Weber number. Various averages of the flow field and the phase distribution, over planes parallel to the walls, are examined and compared, and the microstructure is examined using two-point correlation functions and other measures.

1 Supported by DoE/CASL.

Monday, November 25, 2019 8:00AM - 10:23AM — Session H04 Granular Flows I 203 - Alban Sauret, University of California Santa Barbara

8:00AM H04.00001 Localized rotational effects on granular temperature. JONATHAN HIGHLAND, University of Liverpool, AVINASH VAIDHEESWARAN, WILLIAM FULLMER, NETL, JONAS SAFFON, Polytech Nantes — The granular temperature of a granular flow is seen to be analogous to a the Reynolds stress inside of a fluid flow. In both cases they represent how energy / momentum is passed around the medium. The main difference between a fluid flow and a granular flow is fluid flows physics is governed by its viscosity, whilst a granular flow is governed by collisions and surface roughness. In a recent paper by Higham et al. (2019) it was shown a combination of individual surface roughnesses and collisions can cause rotational moments to be passed between the individual grains. These three dimensional rotations are not typically taken into consideration in simulations or in experiments, but can have quite an effect on the individual energy / momentum fluxes. In this presentation we present an experimental investigation of a two-dimension driven vortex in a granular flow. We use particle tracking to determine the spatial and rotational translations. From these data we determine what effect the localized rotations have of the granular temperature.
8:13AM H04.00002 A Depth-Averaged Method for Estimating Velocity Profile Evolution in a Granular Flow1. BENJAMIN YOUNG, STUART DALZIEL, NATHALIE VRIEND, University of Cambridge — We present a method for estimating the depth-dependent velocity profiles of free surface granular flows from evolving surface velocity and flow-depth data. We first derive a quasi-two-dimensional three-variable depth-averaged model for free surface flows that accommodates the inclusion of a varying basal boundary condition. This model is then used in conjunction with an ensemble Kalman Filter (enKF) and free surface data to examine the inverse problem: What is the internal velocity field of the flow, given our model and free surface data?. We demonstrate the capabilities of this method by applying our algorithm to the Blasius boundary-layer problem described in Tsang et al. (2018). Synthetic, evolving free surface data is generated from discrete particle model (DPM) simulations and then input into the enKF/depth-averaged algorithm in order to estimate the internal velocity. These estimates are then compared to the coarse-grained DPM velocity field and are shown to provide a good fit to the synthetic data. We believe our method has clear potential, not just in modelling free surface flows, but also as a powerful data-analysis tool for experimentally validating modelling work done within the granular community.

1 BAY is supported by a NERC ESS DTP studentship and NMV acknowledges support from a Royal Society Dorothy Hodgkin Fellowship DH120121

8:26AM H04.00003 Using directional-specific shear rates to correlate mass flow rate and velocity profiles in a granular conveyor1. NICHOLAS POHLMAN, HANNAH HIGGINS, MICHAEL ROEING-DONNA, JIFU TAN, Northern Illinois University — Velocity profiles and total mass flow rate of an industrial-style conveyor system with a storage hopper is explored using both experiments and simulation. Despite expectations of quasi-two-dimensional behavior, the velocity profiles observed at the side walls are not consistent throughout the stored material above the flighted conveyor belt. Integrating velocity profiles from high speed imaging of the experiment proved to underestimate the total mass flow rate of the system when different opening sizes and belt speeds were used. Simulations using the LIGGGHTS platform with boundary conditions similar to the experimental parameters confirm that velocity adjacent to the flights may be constant but non-uniform due to the jam-prevention gaps between flights and the walls. Results indicate that the shear rate decays differently in gravitational versus transverse directions. Fitting parameters for two unique shear rates were applied to allow better correlation of the velocity profiles and mass flow rates that were measured.

1 Sponsored by NIU’s Research / Artistry Opportunity Grant

8:39AM H04.00004 Slow granular flows in a Split-Bottom Couette device. PETER DSOUZA, PRABHU NOTT, Indian Institute of Science — We present DEM simulations and experiments of slow shear of granular materials in a split-bottom Couette device. This device is a cylindrical cup with a split base, wherein a central disc that is flush with the bottom rotates, while all other walls of the device are stationary¹. Shear originates at the split between this disc and the rest of the base. This device has been widely used to validate rheological models for granular flows since it exhibits wide shear bands. We show that by changing the fill height of the device, we change the location of shear in the material. We then show the presence of system spanning vortices in addition to the primary azimuthal flow. These vortices are like those seen in the cylindrical Couette device². We show that the form of the vortices can be explained by accounting for shear-induced dilation in the system, validating the arguments made for the vortices seen in the Cylindrical Couette device². No current rheological models for slow granular flow allow for shear-induced dilation and thus can’t capture these vortices. We thus make the case that dilation needs to be included in any rheological model for slow granular flows. 1. Dijksman, & van Hecke, 2010, Soft Matter 2. Krishnaraj, & Nott, 2016, Nature Comm

8:52AM H04.00005 Experimental study of the collapse of cohesion-controlled granular materials1. ALBAN SAURET, UC Santa Barbara, ADRIEN GANS, IUSTI, Aix Marseille Univ, CNRS, MINGZE GONG, UC Santa Barbara, OLIVIER POULIQUEN, MAXIME NICOLAS, IUSTI, Aix Marseille Univ, CNRS — Cohesive granular media are encountered in many geophysical and industrial applications, examples being cement, pharmaceutical powders, or flours. Many progress has been made in the description of dry granular flow, but the flow behavior of powders remains elusive. In particular, one difficulty lies in the cohesion force between the particles. Using a recently developed method to create a Cohesion Controlled Granular Material (CCGM) and relate the interparticle cohesion to the macroscopic behavior, we consider experimentally the collapse of a column made of cohesive grains. This configuration has been extensively studied in the case of dry granular material: when the grains are released, the granular mass spreads and stops at a finite distance. The morphology of the deposit is mainly controlled by the initial aspect ratio and is independent of the material properties. Yet, for powders the inter-particle forces strongly affect the collapse. Here, we characterize the effects of cohesion on the collapse dynamics, the run-out length, and the final morphology of the deposit. These experiments illustrate that cohesive forces between particles introduced an additional complexity in this system.

1This work is part of the COPRINT project and was supported by the ANR grant ANR-17-CE08-0017

9:05AM H04.00006 Energy evolution in the subaqueous granular column collapse process. YI AN, WENTAO ZHANG, Institute of Mechanics, Chinese Academy of Sciences, QINGQUAN LIU, Beijing Institute of Technology — We studied the energy evolution in the subaqueous granular column collapse process experimentally in this work to understand the unusual phenomena of long-runout reported by V. Topin et al. (PRL, 2012). We set up a well-controlled experimental facility with only one granular width to study the one layer subaqueous granular column collapse problem. We obtained the velocity field for both granular and liquid simultaneously by employing the refractive index matching and planar laser-induced fluorescence. We use Hough transform to identify and track the spherical granular and the 2D Particle Image Velocimetry to obtain the velocity field outside the granular body. We find the total kinetic energy $E_k$ of the granular materials drop dramatically when granular particles turn horizontal with a sharp angle, which implies the granular collision is the key factor determining the runout. The viscosity of the ambient liquid may play multiple roles, on one hand, it reduces the total local collision events, on the other hand, it not only dissipates the kinematic energy but also extends the duration of the entire event. Thus a non-dimensional number describing the role of the viscosity is suggested.

This work is part of the COPRINT project and was supported by the ANR grant ANR-17-CE08-0017
9:31AM H04.00008 Bed-load characteristics over evolving and developed subaqueous barchan dunes1, ERICK M. FRANKLIN, CARLOS A. ALVAREZ, UNICAMP - University of Campinas — In the morphodynamics of crescent-shaped dunes, known as barchans, many complex aspects are involved. One of them concerns the trajectories of individual grains over the dune, and how they affect its shape. In this study, we investigate experimentally the formation and evolution of subaqueous barchan dunes in a closed conduit. In our experiments, granular heaps of conical shape were placed on the bottom wall of a rectangular channel and they were entrained by turbulent water flows. We measured the trajectories of grains migrating to horns of both evolving and developed dunes and showed that most of the grains came from peripheral regions upstream of the dune centroid, with significant transverse displacements. These results diverge from the generally accepted description that the barchan horns form from the advance of the lateral dune flanks. Hence, our results reveal a new mechanism for barchan formation that might be complementary to that accepted so far.

1Carlos A. Alvarez is grateful to SENESCYT (Grant No. 2013-AR2Q850) and to CNPq (Grant No. 140773/2016-9). Erick M. Franklin is grateful to FAPESP (Grants No. 2016/13474-9 and No. 2018/14981-7), to CNPq (Grant No. 400284/2016-2) and to FAPEPE/UNICAMP (Grants No. 2210/18 and No. 2112/19) for the financial support provided.

9:44AM H04.00009 Unsteady Shearing of a Granular Material in an Anular Couette Cell1, HAN-HSIN LIN, MELANY HUNT, California Institute of Technology — We study the transition from unsteady to steady state shearing of spherical glass beads and irregular sands in a Couette cell. By initially fluidizing the bed or compressing with a constant force, we ensure the initial state is controlled and repeatable. By comparing to simulations, we are able to capture the structure change inside the bulk. When controlling the torque, the system cannot reach a steady state when it is below a critical stress. When controlling the speed of the boundary, the shear stress at the wall increases slowly over a period of time that depends on the initial state of the bed, wall friction, shear rate, and flow along the free surface. At steady state, the stress decreases at the highest rotation speeds. Simulations with LAMMPS using Hooke's contact model show a recirculation cell driven by gravity and the free surface, which results in the increasing stress observed in the measurements. The relations of wall friction angle to normal stress for different samples have different trends. The effective friction of the inner wall matters. When using the smooth cylinder, the system needs more time to reach a steady state than using the rough cylinder. At steady state, the wall stress decreases more significantly at the highest rotation speeds compared to the rough cylinder.

1This material is based upon work supported by the National Science Foundation under Grant No. 1706166

9:57AM H04.00010 Wiggling arthropods induce flow in granular materials1, KAREN DANIELS, MELIA KENDALL, SHIH-YUAN CHEN, EMILY BROWN, BJORN SUMNER, MICHAEL MANN, North Carolina State University — Just as heating a viscous fluid causes its viscosity to drop, we observe that the introduction of active particles into a passive granular material can increase its flowability. This effect can be observed, for instance, in the historical practice of aging Milbenkase cheese in mixtures of flour and mites. In our experiments, we examine this effect by introducing flour beetle larvae (Tribolium confusum) into agricultural grains of various sizes. We measure the timescale for bulk flow via the relaxation of a sloping pile, and the timescale for particle-scale rearrangements via diffusing wave spectroscopy. We find that the macroscopic and microscopic timescales are approximately proportional; both timescales decrease as the fraction of larvae increases, but only for samples in which the grains are smaller than the larvae. For samples in which the larvae and grains are of similar size, these two timescales decouple.

1NSF DMR-1608097

10:10AM H04.00011 Experimental measurements of the torque and normal force on a helix rotating in a granular material, ROGELIO VALDEZ, Universidad Nacional Autonoma de Mexico, MELANY HUNT, California Institute of Technology, ROBERTO ZENIT, Brown University — The torque and normal force produced by a helix rotating in granular matter were measured experimentally. The experiments were conducted using the rheometer, with a powder cell, for a wide range of rotational speeds. Two granular media were considered: mustard seeds and glass beads with diameter 0.203 mm. The experiments considered changes in the geometry of the helix. For a first set of tests, seven helices with the same total length but with different helix angle and wavelength were considered. For the second group, ten helices with the same geometric shape but with different numbers of turns, from 1 to 9, were used. The results show that torque and normal force are strongly dependent on the helical geometry. A maximum normal force is reached when the helix angle is around 55 degrees while the peak for the torque occurs when the helix angle is close to 40 degrees. In both cases, the measurements are nearly independent on the rotational speed of the helix. Both force and torque increase linearly with the number of coil turns for small number of coils; however, in contrast to what may be expected for a viscous fluid, the increase is not linear when the number of coils is larger than 3. Comparisons with calculations from granular resistive force theory will be presented.
8:00AM H05.00001 The Effects of Parasitic Combustion on Detonation Wave Propagation¹, SUPRAJ PRAKASH², VENKAT RAMAN³, University of Michigan — Rotating detonation engines (RDEs) are a feasible approach to realizing pressure gain combustion. However, practical implementation of these devices requires the use of non-premixed discrete fuel-oxidizer injection. Turbulent mixing of the fuel and oxidizer streams enforces reactant stratification inside the combustor annulus. Parasitic deflagrative combustion of the freshly-injected fuel-oxidizer mixture introduces partially-burnt gases to the detonation wave and diminishes heat release. Numerical studies of full-scale RDE systems have suggested that up to 35-50% pre-burning of the fuel-oxidizer mixture is prevalent within the combustor. The primary objective of this work is to understand the interaction of the detonation wave and a spatially-inhomogeneous mixture. To this end, a high-fidelity numerical simulation approach is utilized to understand how partially-burnt mixtures affect the detonation wave structure by inducing different fractional levels of reactant mixture pre-burning with reference to the fully-burnt equilibrium state. The effect of pre-burning on detonation wave stability and characteristics are discussed to provide insight into the difference between theoretical Chapman-Jouguet detonation and the detonative combustion observed in practical combustors.

1The research is supported by DOE/NELT University Turbine Systems Research under DOE Grant Nos. DE-FE00025315 and DE-FE00023983.

8:13AM H05.00002 Quantification of Detonation Augmentation by Secondary Waves in a Rotating Detonation Combustor¹, FABIAN CHACON, MIRKO GAMBA, University of Michigan — In this work we will investigate the system of waves present in a laboratory scale rotating detonation combustor (RDC). These devices are of scientific interest because of the theoretical efficiency gain that can be achieved through the utilization of these devices in a jet engine or other conventional combustor. However, the flow fields within RDCs are complex and not fully understood, nor are many of the mechanisms behind some of the phenomena associated with a RDC. One such phenomena is termed secondary waves: waves (apart from detonation) which have some associated pressure oscillation, chemical reactions, or both travelling at a consistent speed while the combustor is under operation. In particular, we will use a newly developed analysis technique that allows for the quantification of spatial distributions of pressure throughout the operation of the RDC. This will allow for determining the impact that the secondary wave has on the pressure rise of the detonation wave when they collide in the channel. Understanding how significant the impact of these collisions are, will allow for greater understanding of the role these secondary waves will play into the operability and stability of a RDC and its integration into a practical system.

1This paper is based on work supported by the DOE/NELT University Turbine Systems Research under DOE Grant No. FE0031228 with Robin Ames as program monitor.

8:26AM H05.00003 Effects of non-premixed injection schemes on the detonation structure in rotating detonation engines¹, TAKUMA SATO, VENKAT RAMAN, University of Michigan — Recently, RDEs has been getting more attention as the pressure gain combustor. Because it uses the detonation in the combustion process, the reactant mixture can get additional compression due to the shock wave. Although a series of simulations have been conducted in the community, most of them are limited in the canonical problems and the premixed assumption in a 2D geometry. In the real RDEs system, the non-premixed injection system creates a complex detonation structure due to the incomplete mixing and the stratification of the fuel and oxidizer. However, the measurement of the detailed flame structure is hard to obtain in the experiment due to the harsh environment of the system. With this mind, the goal of this study is to understand the detailed 3D detonation structure by simulating the full system RDEs system. Because the flow-field is highly unsteady in time and space, the 3D averaged flow-field will be extracted from the simulation. The heat release distribution in the space will be extracted to understand the combustion process in the detonation chamber. The mixing process of the non-premixed injection system will be discussed by varying the mass flow rate. Finally, the comparison between the Euler and Navier-Stokes equations will be discussed.

1DOE-NELT DE-FE00025315, DOE-NELT DE-FE00023983

8:39AM H05.00004 Analysis of mode transition in Rotating Detonation Engines using detailed numerical simulations, PRASHANT TAREY, PRAVEEN RAMAPRABHU, University of North Carolina, Charlotte, JACOB MCFARLAND, University of Missouri, Columbia, DOUGLAS SCHWER, Naval Research Laboratory, Washington DC — Detonation Engines (RDE) can operate in single or multiple detonation wave modes, while the mode of operation depends on several factors including the equivalence ratio, mass flow rate etc. In this work, we analyze the mechanism of mode transition through detailed numerical simulations of a 2D unrolled RDE geometry with discrete injectors. We systematically vary the equivalence ratio of the hydrogen-air mixture in our simulations with 1-step chemistry. The different modes of operation and the parameter boundaries separating them, were investigated and compared with experimental¹ results. Our results show that the number of waves is proportional to the equivalence ratio as well as the detonation cell width. The effect of the detonation modes on thrust and detonation height were also investigated. The compressible Euler simulations were solved on a Cartesian grid with Adaptive Mesh Refinement, using the Piecewise Parabolic Method (FLASH²), while a second-order accurate, Immersed Boundary Method was implemented to model the discrete injectors. ¹A. George et al., Proc. Comb. Inst., 36 (2), 269L. (2017). ²B. Fryxell et al., Astrophys. J., Suppl. Ser. 131, 273 (2000).

8:52AM H05.00005 ABSTRACT WITHDRAWN —

9:05AM H05.00006 Numerical Investigation of the Accuracy of Particle Image Velocimetry Technique in Gas-Phase Detonations, SAI SANDEEP DAMMATI, YORAM KOZAK, Texas A&M University, KAREEM AHMED, University of Central Florida, ALEXEI POLUDNENKO, Texas A&M University — In this study, we numerically investigate the accuracy of the Particle Image Velocimetry (PIV) technique for the flow characterization in high-speed, compressible regimes, and in particular in gas-phase detonations. A two-dimensional, planar detonation at atmospheric conditions is modeled using a simplified single-step Arrhenius kinetics. The upstream flow is uniformly seeded with particles representing the Al2O3 PIV particles used in experiments, along with initially co-located massless Lagrangian tracers used to recover the correct particle trajectories in the flow field. Massless Lagrangian particles are integrated using both 2nd order and 4th order time integrators to further assess the errors in the reconstructed Lagrangian trajectories. By comparing the trajectories of massive particles with those of the tracer particles, we address the following questions: a) How do PIV particles affect the detonation wave, in particular its velocity and cellular structure? b) How closely do massive PIV particles follow the flow pathlines? c) What is the accuracy of the flow field reconstructed using the PIV particles? Finally, we discuss the implications for the use of the PIV technique as a diagnostic tool for high-speed reacting flows such as detonations in detonation-based engines.

¹Department of Defense High Performance Computing Modernization Program
Scramjets with Hydrogen, Methane, and Ethylene Fuels, WENHAI LI, TTC Technologies, Inc., LADEINDE FOLISIO, Department of Mechanical Engineering, Stony Brook University — A few results on flame stability and thermal choking in a simplified model of the scramjet engine will be presented for hydrogen, methane, and ethylene fuels which are injected in crossflow to a supersonic airflow in a combustor. The parameter space investigated includes a range of air stagnation temperatures ($T_0$), jet-to-freestream momentum flux ratios ($J$), and the three fuels. The analysis is done via the large-eddy simulation and within the context of the flamelet approach. Preliminary results show that hydrogen and ethylene have similar thermal choking limits in terms of $T_0$ and $J$, while the methane flames are quite difficult to maintain under the current test conditions.

Modeling of the Cellular Structure of Detonation in Liquid Explosives, LUKE EDWARDS, University of Arizona, MARK SHORT, Los Alamos National Laboratory — Detonation waves propagating in some liquid explosives such as nitromethane (NM) are known to exhibit complex cellular patterns reminiscent of those observed in gaseous explosives. Such cellular structures in NM/diluent mixtures have been recorded by framing camera images (Fickett and Davis 1979). The origins of such instabilities are disputed, ranging from hydrodynamically generated instabilities to failure waves propagating into the detonation reaction zone from the NM/confiner boundary. Detonation front shapes measurements by streak camera imaging in NM/diluent mixtures, on the other hand, mostly show smoothly curved shock fronts, which appear to contradict the observed presence of cellular instabilities. We also know that detonations in NM mixtures are carbon rich, which results in a spatially elongated zone of carbon coagulation behind a much thinner main reaction layer. In this work, we examine the potential origin of the observed cellular detonation instabilities in NM mixtures via an asymptotic theory that explicitly accounts for the long carbon coagulation region. The theory explores why the presence of cellular instabilities may not manifest themselves on the detonation shock front.

Steady Detonation Propagation in Thin Channels with Strong Confinement, MARK SHORT, STEPHEN VOELKEL, CARLOS CHIQUETE, Los Alamos National Laboratory — We examine asymptotically the dynamics of 2D steady detonation wave propagation and failure for a strongly confined high explosive (HE), where the width of the explosive is small relative to the reaction zone length. An energy balance equation is derived which shows how the longitudinal acceleration of subsonic flow behind the detonation shock is influenced both by chemical reaction and by the effects of HE boundary streamline deflection, specifically via the induced rate of change of mass flux through the detonation wave. The latter serves to either counteract or reinforce the acceleration of longitudinal flow depending on the gradient of the boundary streamline deflection. The analysis is valid for general equations-of-state and chemical reaction rates in the HE. The energy equation represents an eigenvalue problem for the detonation phase speed. We explore specific results for the ideal- and stiffened-gas equations of state, along with a pressure-dependent reaction rate for which changes in the pressure exponent and reaction order are also studied. We consider the influences of both straight and curved HE boundary streamline shapes. The asymptotic analysis reveals significant physical insights into how detonation propagation and failure is affected by strong confinement.

The effect of the thermodynamic closure on shock-to-detonation transition modeling in condensed-phase high explosives, CARLOS CHIQUETE, MARK SHORT, STEPHEN VOELKEL, Los Alamos National Laboratory — The need for accurate prediction of detonation initiation and propagation in high explosives (HEs) has lead to various empirical constitutive models for the HE’s equation of state (EoS) and reaction rate. These experimentally calibrated models are used at the continuum level where it is possible to efficiently calculate detonation wave motion at the engineering scale. A transition from the (solid) reactant to the (gas) product state occurs via a single irreversible reaction, requiring a closure condition between the two phases which must then coexist in a single material element. Different closures have been used in the past, for example, pressure-temperature equilibrium. However, analysis of more physical, explicitly 2-phase modeling approaches where each phase’s thermodynamic state can evolve from the (solid) reactant to the (gas) product state occurs via a single irreversible reaction, requiring a closure condition between the two phases which must then coexist in a single material element. Different closures have been used in the past, for example, pressure-temperature equilibrium. However, analysis of more physical, explicitly 2-phase modeling approaches where each phase’s thermodynamic state can evolve have shown that temperature equilibration occurs over a much longer time scale than the corresponding reaction zone scale. Nevertheless, this “nonphysical” closure has been shown to capture the shock-to-detonation-transition (SDT) for many HEs. To clarify this, we systematically vary the closure condition and isolate its effect by fixing the EoS models and reaction rate form. Computational results using the various closures will then be compared and contrasted with respect to SDT data.

Analysis of Detonation Driving Zone in Condensed-Phase High Explosives at Varied Confinements, STEPHEN VOELKEL, CARLOS CHIQUETE, MARK SHORT, Los Alamos National Laboratory — Lateral yielding of confiners used in detonations of condensed-phase high explosives (HEs) introduces streamline divergence into the flow and subsequently reduces the detonation’s phase velocity to below its planar Chapman-Jouguet (CJ) limit. The resulting phase velocity is determined by the subsonic flow region behind the shock front, denoted the detonation driving zone (DDZ). The DDZ itself is dependent on the HE material properties, the thickness of the charge, and the confiner properties. Holding all else equal, at infinite thickness the phase velocity approaches the CJ limit. As the thickness decreases, so does the phase velocity until the thickness reaches some critical value, below which the detonation is unable to sustain itself. In this work, we consider an idealized HE material and simulate steady detonations over a wide range of confinements. For each confinement, simulations at varied thicknesses between the CJ and critical limits are performed and analyzed. Correlations of the DDZ with the phase velocity are presented, with a specific focus on structures and relations that are independent of the confinement. Furthermore, we show that properties within the DDZ correlate with the critical thickness.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H06 Reacting Flow Instabilities and Near-Limit Flames 205 - Carlos Pantano, University of Southern California
8:00AM H06.00001 A Computational Study on Chemical Characteristics of Bluff-body-stabilized Lean Premixed Flames1. YU JEONG KIM, WONSIK SONG, FRANCISCO E. HERNANDEZ PREZ, King Abdullah University of Science and Technology (KAUST), BOK JIH LEE, Seoul National University, HONG G. IM, King Abdullah University of Science and Technology (KAUST), KAUST TEAM — Bluff-body flame holders have been used to achieve stable combustion by recirculating hot product gases behind the holders in premixed reacting flow systems under highly unstable conditions. Despite assisting to improve combustion stability, holders also induce instabilities to flames and flow fields such as vortex shedding and blow-off. In the present study, high-fidelity direct numerical simulations are conducted to investigate flame dynamics behind a bluff-body in lean premixed mixtures and to understand the flame stabilization mechanism, in particular under hydrodynamic instabilities. A three-dimensional rectangular channel with a square-section bluff-body flame holder is selected as configuration. Several distinct flame instabilities are identified as the blow-off limit is approached. The sequence of key physical mechanisms, such as extinction and re-ignition, and combustion and chemical characteristics of the flames are also examined using the combined approach of computational singular perturbation and tangential stretching rate.

1This work was sponsored by King Abdullah University of Science and Technology (KAUST) and computing cluster provided by KAUST Supercomputing Laboratory (KSL).

8:13AM H06.00002 Effect of compressibility on laminar flame speed and its influence on the Darrieus-Landau instability of a planar front of premixed flame1. YASUHIDE FUKUMOTO, Institute of Mathematics for Industry, Kyushu University, KEIGO WADA, Center of Coevolutionary Research for Sustainable Communities, Kyushu University, SNEZHANA ABARZHI, The University of Western Australia — The effect of compressibility on a premixed flame front is investigated by use of the method of $M^2$ expansions for small Mach numbers. We study the inner structure of the reaction layer, by applying the method of matched asymptotic expansions to an overall one-step irreversible chemical reaction expressed by the Arrhenius law. We figure out the structure of freely propagating deflagration wave from the perspective of the temperature distribution and the laminar flame speed. The temperature distribution is greatly influenced by the compressibility effect which originates from the material derivative of the pressure in the source term of the heat-conduction equation. This effect sensitively drops down by the compressibility effect. This implies that the compressibility acts to reduce the growth rate of the Darrieus-Landau instability.

1This work was supported in part by a Grant-in-Aid for Scientific Research from the Japan Society for the Promotion of Science (Grant No. 19K03672).

8:26AM H06.00003 Gravity effects on the nonlinear dynamics of premixed flames. CHRISTOPHE ALMARCHA, BASILE RADISSÖN, BRUNO DENET, Aix Marseille Université, CNRS, Centrale Marseille, IRPHE UMR 7342, 13384 Marseille, France — During their propagation, premixed flames are unstable and undergo a rich dynamics that can be favorably compared to the integration of Michelson-Sivashinsky (MS) equation. Three main ingredients are involved in this nonlinear partial differential equation: the Darrieus-Landau non-local instability, the normal propagation of the front and a small scale diffusive stabilization. In order to take into consideration the long range effects of gravity, we study a modified (MS) equation where a linear stabilizing term is added. We demonstrate that this modified equation is capable of reproducing features of experimental 2D flames propagating downwards in a Hele-Shaw burner. In particular, although the additional term is linearly stabilizing, it is responsible for an increase in the fragmentation of the interface into smaller cells.

8:39AM H06.00004 Nonlinear stability of a premixed flame subjected to a transverse shear2. XIAOYI LU, MOSHE MATALON, University of Illinois at Urbana Champaign, CARLOS PANTANO, University of Southern California — We present our findings on the hydrodynamic stability behavior of a premixed flame subjected to a transverse shear. The analysis is carried out in the weak thermal expansion limit resulting in a modified Michelson-Sivashinsky (MS) equation, which describes the evolution of the flame surface. Numerical solutions to the MS equation show that due to the transverse shear, the flame develops a skewed cusp-like flame, that steadily propagates into the unburned gases and simultaneously translates along the transverse direction. Both propagation and translation velocities are shear-dependent. The fully nonlinear evolution of premixed flames with a realistic density jump at higher shear levels is then investigated using the Direct Numerical Simulations approach.

8:52AM H06.00005 Flow Field Measurements for Instability Mitigation in Lean Direct Injection3. JOSHUA KRSEK, Rose-Hulman Institute of Technology — Lean direct injection (LDI) is a method of combustion used in aviation gas turbines which results in reduced emission of carbon monoxide, nitric oxides (NOx), soot, and unburned hydrocarbons. LDI often results in potentially destructive thermoacoustic instabilities, which can lead to combustor structural failure. A ceramic foam insert has been shown to passively mitigate such thermoacoustic instabilities. Further work has shown that this effect is caused at least in part by the porosity of the insert, as fully dense inserts are not universally effective at mitigating these instabilities. This study attempts to develop a fundamental understanding of the combustor flow structure by analyzing particle image velocimetry (PIV) for combustion under the presence of a porous insert, solid insert, and no insert. The results from this study will play a pivotal role in the design of an optimal instability-weakening porous insert for the NASA developed swirler-venturi injector.

1University of Alabama REU site. NSF grant 1659710. NASA grant NNX13AN14A.

9:05AM H06.00006 Acoustically-Coupled Combustion Dynamics of Laminar Microjet Diffusion Flames4. ANDRES VARGAS, JOSE GUERRERO, ANN KARAGOZIAN, HYUNG SUB SIM2, University of California, Los Angeles — The present experiments focus on the response of burning gaseous fuel jets to prescribed transverse acoustic excitation as a means of exploring the coupling of reactive, acoustic, and flow processes relevant to combustion instabilities. Microjets with several alternative configurations and sizes (including single and multiple jets) and different fuels are exposed to transverse standing wave disturbances within an acoustic waveguide for which a range of resonant frequencies and amplitudes of excitation are applied. Temporal flame response to acoustic excitation is studied via OH* chemiluminescence and visible imaging, with quantification of the dynamics via proper orthogonal decomposition (POD), Rayleigh indices, and temporal flame distortion measurements. Characteristic signatures associated with various types of flame response are identified, including weakly oscillatory combustion, full-scale flame lock-in to excitation, and multi-mode flame dynamics preceding flame extinction. Phase space plots of dominant POD mode coefficients produce periodic as well as strange attractor-like shapes for high amplitude forcing.

1Supported by AFOSR Grants FA9550-15-1-0339 and FA9550-19-1-0095 (PO: Dr. Mitat Birkan)
2Currently with Sandia National Laboratories, Combustion Research Facility
9:18AM H06.00007 Theoretical Source Statistics of Thermo-Acoustic Instability within Reacting Flow. STEVEN MILLER, University of Florida, Department of Mechanical and Aerospace Engineering. We develop a statistical tool based on the equations of motion to quantify the various acoustic sources that cause thermo-acoustic instability. The Navier-Stokes equations, energy equation, and mass fraction equations are decomposed into their base, aerodynamic, and acoustic components. The acoustic components are solved for via the method of the vector Green’s function. The spectral densities of acoustic fluctuations are then formed. The resultant two-point cross-correlations of the Navier-Stokes operator, energy equation operator, and mass fraction operator on the resolved aerodynamic flow-field yield thermo-acoustic statistical sources. Resultant sources are the two-point cross-correlations consisting of terms involving traditional aerodynamic interactions, aerodynamic-combustion interactions, and combustion-combustion interactions. We show that under certain simplifying assumptions, the traditional Rayleigh criterion and combustion acoustic equations are recoverable. We compare the newly proposed theory to traditional Rayleigh criterion and acoustic equations. The benefit of the theory is all thermo-acoustic sources are accounted for in a single two-point cross-correlation model, which is consistent with traditionally accepted theory.

9:31AM H06.00008 Computational Investigation of Combustion Instability in a Gas Turbine Model Combustor. NICHOLAS ARNOLD-MEDABALIMI, CHENG HUANG, KARTHIK DURAISAMY, University of Michigan. Gas combustion is expected to see continued use in propulsion and energy applications for the foreseeable future. Lean operating conditions are desired from an emissions standpoint but can lead to thermoacoustic instabilities. Gas Turbine Model Combustors (GTMC) have been investigated in recent years to improve our understanding of the underlying phenomena. A well-examined experimental design is that of Meier et al. investigated both at DLR and the University of Michigan. In this work, we perform reacting Large Eddy Simulations (LES) of this partially premixed swirl stabilized burner at various operating conditions with particular interest in thermoacoustically unstable regimes. Detailed validations are presented with experiments, and time-averaged field characteristics, acoustics, and coherent structures are examined. The impact of turbulent combustion closure models is examined by comparing the performance of finite-rate chemistry and flamelet regimes. Detailed validations are presented with experiments, and time-averaged field characteristics, acoustics, and coherent structures are examined. The impact of turbulent combustion closure models is examined by comparing the performance of finite-rate chemistry and flamelet regimes.

9:44AM H06.00009 Temporal linear stability analysis of laminar flames on inclined fuel surfaces. RAQUEL HAKES, University of Maryland, WILFRIED COENEN, ANTONIO SANCHEZ, University of California, San Diego, MICHAEL GOLLNER, University of Maryland, FORMAN WILLIAMS, University of California, San Diego. Experiments have found substantial structural differences between buoyancy-driven flames developing on the upper and lower surfaces of inclined burning plates. These differences cannot be explained with existing analytical solutions of steady semi-infinite flames, which provide identical descriptions for the top and bottom configurations. We perform a temporal linear stability analysis to investigate the potential role of flame instabilities in the experimentally observed flow differences. The problem is formulated in the limit of infinitely fast reaction, considering the non-unity Lewis number of the fuel vapor. The analysis incorporates nonparallel effects of the base flow and considers separately spanwise traveling waves and Görtler-like streamwise vortices. The solution to the stability problem determines the downstream location at which the flow becomes unstable, characterized by a critical value of the local Grashof number, which varies with the plate inclination angle. The results for the underside flame indicate that instabilities emerge further downstream than they do for a topside, in agreement with experimental observations. Increased baroclinic vorticity production is reasoned to be responsible for the augmented instability tendency of topside flames.

1Based upon work supported by the NSF Graduate Research Fellowship Program Grant No. DGE 1840340 and NSF Award No. 1554026; RH supported by the Clark Doctoral Fellowship Program.

9:57AM H06.00010 Near-limit H₂-O₂-N₂ combustion in nonpremixed counterflow mixing layers. PRABAKARAN RAJAMANICKAM, University of California San Diego, JAIME CARPIO, Universidad Politecnica de Madrid, ANTONIO L. SANCHEZ, University of California San Diego, PAUL D. RONNEY, University of Southern California, FORMAN A. WILLIAMS, University of California San Diego. Numerical computations employing detailed chemistry and experiments in a slot-jet counterflow burner are used to characterize the different combustion modes emerging in mixing layers separating N₂-diluted counterflowing planar streams of hydrogen and oxygen. Attention is focused on high degrees of dilution resulting in near-critical flames with peak temperatures close to the crossover temperature. A bifurcation diagram is presented in the mixture-fraction vs. strain-rate plane that identifies six different combustion regimes involving four different flame types, namely, diffusion-flame sheets, propagating RETREATING edge flames, broken-flame tubes, and isolated flame tubes. While the isolated flame tube is always stationary, the broken flame tubes can be stationary or can propagate with an oscillatory speed. The observed flame behavior exhibits hysteresis in some parametric regions, where the flow that is established depends on the ignition mechanism.

10:10AM H06.00011 Quantification of local combustion modes in laminar non-premixed n-dodecane/air coflow flames. CHAO XU, SIBENDU SOM, Argonne National Laboratory. Non-premixed lifted jet flames in heated air coflow are important to a range of practical propulsion systems, while stabilization mechanisms for complex fuels have not been fully understood. To better understand the structure and stabilization mechanisms, two-dimensional laminar n-dodecane jet flames in heated air coflow at 30 atm are simulated with detailed chemical kinetics. To quantify roles of different sub-processes (e.g., chemistry, diffusion, radiation, etc.) and associated local combustion modes, a new flame diagnostic based on chemical explosive mode (CEMA) analysis is developed, by projecting local chemical and diffusion source terms to the direction of CEM and quantifying the competition between these terms. Diffusion processes in both normal and tangential directions of the non-premixed flame sheet (defined as the mixture fraction isocontours) are further accounted for. Additional local combustion modes specific to nonpremixed flames (compared to premixed flames) are identified, highlighting different roles of flame-normal and flame-aligned diffusion processes in stabilizing the flames. Effects of radiative loss on local combustion modes and the flame stabilization mechanism, are also discussed based on an optically-thin radiation model.

10:23AM H06.00012 Numerical study on the stabilization of ball-like reacting fronts at normal gravity. FRANCISCO HERNANDEZ PEREZ, King Abdullah University of Science and Technology, ZHEN ZHOU, YURIY SHOSHIN, JEROEN VAN OIJEN, PHILIP DE GOEY, Eindhoven University of Technology, HONG IM, King Abdullah University of Science and Technology. A computational investigation on the stabilization and dynamics of lean premixed flames in tubes at near lean-limit conditions is conducted. Hydrogen-methane-air premixed mixtures are considered in tubes with two different diameters (13.5 and 55 mm) and under the influence of normal gravity. As the lean flammability limit is approached, individual (13.5 mm tube) and multiple (55 mm) ball-like flames are formed and stabilized. Through high-fidelity simulations with detailed transport and chemical models as well as the inclusion of heat losses, the stabilization and dynamics of such flames are examined.

The support from KAUST, the KAUST Supercomputing Laboratory and the Dutch Technology Foundation (STW) are gratefully acknowledged.
8:00AM H07.00001 On particle distribution in High-Order Smoothed Particle Hydrodynamics schemes, RENATO VACONDIO, university of parma — Smoothed particle hydrodynamics (SPH) numerical schemes are becoming very popular in CFD due to their Lagrangian and meshless nature. Nevertheless, many arising numerical issues still remain to be investigated. One of the major drawbacks of such meshless schemes is the low order convergence rate due to particle spatial anisotropy. To overcome this, considerable work has been done towards introducing kernel corrections. Nevertheless, these might lead to instabilities and break the conservation properties of the numerical scheme. Recently, it was demonstrated that schemes such as diffusion-based particle shifting are able to improve the accuracy of the approximations however, they do not conserve linear and angular momentum. In the present work, an arbitrary Lagrangian-Eulerian scheme (ALE-SPH) has been developed where the transport velocity is computed by means of an iterative particle shifting scheme which ensures a near isotropic particle distribution spatially. Moreover, a new class of kernels have been adopted that can guarantee an arbitrary order of convergence (at the continuous level of spatial interpolation). In this way, we have been able to attain the theoretical order of convergence of the adopted kernel (for example 4th or 6th order) preserving the conservation properties.

8:13AM H07.00002 Modeling Astrophysical Transients with Smooth Particle Hydrodynamics, CHRIS FRYER, Los Alamos National Laboratory — Mesh free methods, and smooth particle hydrodynamics in particular, have been used extensively in modeling astrophysical transients. The ability of SPH to conserve angular momentum, easily include detailed microphysics, and to resolve the mass over large distance scales makes it an ideal method for many problems. I will present an overview of the wide variety of astrophysical transients modeled with smooth particle hydrodynamics, including supernovae, gamma-ray bursts and merging neutron stars. I will focus on the specific strengths and current issues using these methods to model these problems.

8:26AM H07.00003 Incompressible SPH and New Developments, STEVEN LIND, University of Manchester, SPH@MANCHESTER TEAM — SPH shows considerable promise for modelling a range of challenging fluid phenomena, especially those involving highly deforming free-surface flows or interfaces with potential topology change. SPH is often applied in weakly compressible form, but incompressible SPH is a newer alternative gaining popularity given its ability to predict accurate pressure fields without the use of empirical equations of state, artificial sound speeds, or excessive numerical diffusion. This talk will provide an overview of recent developments for the incompressible SPH approach, including examples of its application in quite diverse areas of fluid mechanics. New methodologies for achieving stability and very high accuracy will also be discussed.

8:39AM H07.00004 A spatially adaptive high-order meshless method for fluid-structure interactions, WENXIAO PAN, WEI HU, University of Wisconsin-Madison, NATHANIEL TRASK, Sandia National Laboratories — We present a scheme implementing an a posteriori refinement strategy in the context of a high-order meshless method for problems involving point singularities and fluid-solid interfaces. The generalized moving least squares (GMLS) discretization used in this work has been previously demonstrated to provide high-order compatible discretization of the Stokes and Darcy problems, offering a high-fidelity simulation tool for problems with moving boundaries. The meshless nature of the discretization is particularly attractive for adaptive h-refinement, especially when resolving the near-field aspects of variables and point singularities governing lubrication effects in fluid-structure interactions. We demonstrate that the resulting spatially adaptive GMLS method is able to achieve optimal convergence in the presence of singularities for both the div-grad and Stokes problems. Further, we present a series of simulations for flows of colloid suspensions, in which the refinement strategy efficiently achieved highly accurate solutions, particularly for colloids with complex geometries.

8:52AM H07.00005 Projection Particle Methods - Latest Advances and Future Perspectives, ABBAS KHAYYER, Associate Professor, Kyoto University, Japan — This talk comprises of a review on the latest achievements made in the context of projection particle methods. Projection particle methods, including Incompressible Smoothed Particle Hydrodynamics (ISPH) and Moving Particle Semi-implicit (MPS), are founded on Helmholtz-Leray decomposition and its corresponding Chorin’s projection method. In this talk this important mathematical concept and related mathematical conditions will be reviewed and its application for ISPH and MPS will be concisely described. Followed by providing the mathematical background, the ISPH and MPS numerical methods will be briefly introduced. The latest achievements corresponding to stability, accuracy and energy conservation enhancements as well as advancements related to simulations of multi-phase flows and hydroelastic fluid-structure interactions will be discussed. In specific, more attention will be dedicated to hydroelastic fluid-structure interactions in the context of projection particle methods, introducing methods for interactions of incompressible fluid flows with elastic structures in both Newtonian and Hamiltonian frameworks. Finally, the future perspectives for further enhancements of applicability and reliability of particle methods will be highlighted.

9:05AM H07.00006 Multiscale SPH model for multiphase flow, AMANDA HOWARD, Pacific Northwest National Laboratory — We present nonlocal multiscale partial differential equations for multiphase flow and their smoothed particle hydrodynamics (SPH) discretization. We demonstrate that this model is able to describe multiphase flow at scales ranging from micro (nano) to macro scales and predicts curvature dependence of the surface tension. The nonlocal model is obtained in the form of an integral of a molecular-force-like function added into the momentum conservation equation. Our SPH simulations of multiphase flow in porous materials and droplet and film flow on rough surfaces further reveal multiscale features of the proposed model.

1This work was supported by the US Department of Energy through the Los Alamos National Laboratory.

1We acknowledge support from DOE ASCR.
9:18AM H07.00007 Anisotropic dispersion with a consistent smoothed particle hydrodynamics scheme

We acknowledge funding from the European Union’s Horizon 2020 Programme under the ENERXICO Project agreement grant No. 828947 and the Mexican CONACYT-SENER-Hidrocarbores No. B-S-69926.

9:31AM H07.00008 Hydrodynamic Study of an Extraterrestrial Ship Design with SPH

We report results and lessons learned from a study in which three Smoothed Particle Hydrodynamics (SPH)-methods were used for the solution of several fluid-solid Interaction problems. The first SPH approach considered is widely used in the community and it relies on an equation of state to weakly enforce compressibility (called WCSSP, from Weakly Compressible SPH). The second SPH approach uses an implicit solution in which the incompressibility is enforced by a pressure distribution computed via a Poisson equation (called ISPH, from Implicit SPH). Lastly, a constraint-based method is considered that enforces incompressibility by imposing the constant density condition via a kinematic holonomic constraint on the motion of the SPH markers (called KCSPH, from Kinematically Constrained SPH). WCSSP, ISPH and KCSPH are compared in conjunction with five tests: an incompressibility benchmark test, Poiseuille flow, flow around cylinder, dam break, and fluid sloshing. The interest is in comparing solution attributes such as accuracy, robustness, efficiency, and ease of use and implementation. Ultimately, this effort provided a mechanism to probe the agency of the SPH method.

9:44AM H07.00009 A comparison of three SPH methods for the solution of the Fluid-Solid Interaction problem

A thank you to CONACYT for the financial support for this work.

9:57AM H07.00010 GPUSPH modeling of waves and currents in the nearshore

One billion particles and beyond. In these cases, one can contemplate switching to continuum models for granular dynamics. Inspired by the fluid-like behavior of granular material, in this contribution we report on an approach in which granular material dynamics is approximated via a drodynamics scheme. For the fluid flow, we employ the Smoothed Particle Hydrodynamics (SPH) to solve the Navier-Stokes equations. Similarities between the discrete, fully resolved model and the continuum granular material model are reported drawing on a set of three examples: compressibility test, the classical dam break problem, and the dam break simulation with an obstacle.

10:10AM H07.00011 Granular dynamics vs. fluid dynamics: similarities and differences

As shown by Dalrymple and Iribarren (JGR, 1975), synoptic scale wind waves lead to the formation of nodal and anti-nodal lines in the water surface by superposition. This leads to the formation of circulation cells in the nearshore with rip currents flowing offshore on the nodal lines. However, this is only true for small amplitude waves as nonlinear amplitude diffraction leads to waves crossing the linear nodal lines. This leads to isolated wave crests (Wei et al., 2017) and an increase in the number of waves in the surf zone. This nonlinear effect leads to additional circulation cells within the surf zone. We also examine the magnitude of each term in the wave-averaged equation of motion spatially.

1 Office of Naval Research

10:23AM H07.00012 Numerical simulation of the fluid-structure interaction in an airship through the SPH methodology

The Lagrangian methodology is used to study the stability of the Rolling, Pitching and Yawing moments, and the drag and lift average aerodynamic coefficients. It is assumed that the airship is flying in a low Reynolds number regime in a quasi-compressible medium (air). The numerical results are compared with previous studies, both experimental and numerical simulations using the traditional Eulerian-mesh based methods.

1 A thank you to CONACYT for the financial support for this work
8:00AM H08.00001 Indentation into a plastic fluid layer, THOMASINA BALL, NEIL BALMOUTH, University of British Columbia, IAN HEWITT, University of Oxford — We study the indentation of a rigid object into a layer of a cohesive or non-cohesive plastic material. Existing approaches to this problem using slip-line theory assume that the penetration depth is relatively small, employing perturbation theory about a flat surface. Here, we use two alternative approaches to account for large penetration depths, and for the consequent spreading and uplift of the surrounding material. For a viscoplastic fluid, which reduces to an ideal plastic under the limit of vanishing viscosity, we adopt a viscoplastic version of lubrication theory. For a Mohr-Coulomb material, we adopt an extension of slip-line theory between two parallel plates to account for arbitrary indenter shapes. We compare the theoretical predictions of penetration and spreading with experiments in which a flat plate, circular cylinder or sphere are indented into layers of Carbopol or glass spheres with successively higher loads. There is a clear layer-depth dependence of the indentation and uplift for the viscoplastic material, with a much weaker dependence on layer depth for a Mohr-Coulomb material. Understanding the dynamics of indentation into a viscoplastic layer is of particular importance in the formation of footprints, either by animals or in an industrial process.

8:13AM H08.00002 Dynamics of viscoelastic ribbons, NEIL BAMFORTH, University of British Columbia, IAN HEWITT, Oxford University — A reduced model is presented for the dynamics of slender ribbons of viscoelastic fluid. The model is used to explore the effects of viscoelasticity on the dynamics of a curling ribbon, a drooping cantilever, buckling sheets, snap-through and a falling catenary.

8:26AM H08.00003 Jeffery orbits in shear-thinning fluids, ARMAN ABDIH, GWYN ELFRING, University of British Columbia — At zero Reynolds number, spheroids and long slender bodies in shear flow undergo a periodic motion. In the absence of inertial and Brownian forces, the motion of a neutrally buoyant ellipsoid of revolution in a simple uniform shear was solved by Jeffery who found that the particles axis of revolution rotates on infinitely many degenerate periodic orbits called “Jeffery orbits”. Since this classical work, a number of various studies have demonstrated that inertia (both of the particle and the fluid) and viscoelasticity tend to have a dramatic effect on the particle dynamics, but the effects of shear-dependent viscosity has not previously been explored. In this talk we consider the dynamics of a neutrally-buoyant prolate spheroid in a shear flow of a shear-thinning fluid. We model the fluid using a Carreau model and use perturbation theory about a flat surface. Here, we use two alternative approaches to account for large penetration depths, and for the rheological properties of the shear-thinning liquid and the characteristics of the impacting object. We find that the symmetry of the Jeffery orbits in Newtonian fluids is maintained when the fluid displays shear-thinning, however the changes in viscosity tend to increase the time the particle spends aligned with the flow.

8:39AM H08.00004 Numerical Study of the Flow of a Shear-Thinning Fluid Past a Circular Cylinder Forced to Oscillate Sinusoidally1, UMANG PATEL, JONATHAN ROTHSTEIN, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — Current study numerically investigates two-dimensional laminar flow of shear-thinning fluids past a circular cylinder forced to oscillate in crossflow direction. Merged PISO-SIMPLE algorithm (PIMPLE) has been used to solve the governing equations on unstructured grid. The sinusoidal oscillations of cylinder were handled by solving cell-centre laplacian for mesh motion displacement. Oscillation frequency has been kept in such a way that a reduced velocity remains close to 6. The shear-dependent viscosity has been modelled by the Carreau model where power index is kept below unity to model shear thinning fluid. Reynolds number is defined based on zero shear-rate viscosity. Vortex shedding is observed at very low Reynolds number as compared with the Newtonian fluid since shear-thinning effects are causing the flow to destabilize. For various values of Reynolds number and Carreau model parameters, lift and drag coefficients as well as time- averaged normalized viscosity have been reported. As opposed to Newtonian fluids, we observed the decrease in mean drag value with increasing Reynolds number due to shear thinning effects. The forced oscillations result in shedding of vortices at Reynolds numbers lower than the critical Reynolds number to observe shedding in a fixed cylinder.

1National Science Foundation CBET-1705251 (NSF)

8:52AM H08.00005 Diving into a shear-thickening bath, PHILIPPE BOURRIANNE, Mechanical Engineering, MIT, ROBERT E. COHEN, Chemical Engineering, MIT, GARETH H. MCKINLEY, Mechanical Engineering, MIT — Shear-thickening fluids, made of suspensions of micro or nanoparticles, react to imposed excitations with a tunable behavior. At low shear-rate, they flow like a Newtonian or weakly shear-thinning liquid, whereas their viscosity rapidly increases following a more rapid perturbation. Due to this enhanced dissipation, shear-thickening fluids are known for their remarkable ability to absorb energy during collisions. When a solid object impacts a bath of shear-thickening fluid, the initial velocity determines the different settling regimes that are observed. We will describe these different regimes with regard to the rheological properties of the shear-thickening liquid and the characteristics of the impacting object. A few surprising observations could be noticed. First, a high velocity is not always the best way to penetrate such suspensions. Under such conditions, an appropriately-shaped fast-moving object can also bounce during the impact due to the shear-thickening behavior. By comparing the deceleration of an object into a viscous Newtonian and a shear-thickening liquid, we will explain the spectacular properties of shear-thickening during a collision.

9:05AM H08.00006 Dynamic Wetting Failure in Shear-Thinning and Shear-Thickening Liquids, VASILEIOS CHARITATOS, WIESLAW SUSZYNSKI, Department of Chemical Engineering and Materials Science, University of Minnesota, MARCIO CARVALHO, Department of Mechanical Engineering, PUC-Rio, SATISH KUMAR, Department of Chemical Engineering and Materials Science, University of Minnesota — Dynamic wetting failure in shear-thinning and shear-thickening liquids is examined in this work. Flow visualization experiments using a curtain-coating geometry suggest that shear thinning postpones the onset of wetting failure and the resulting air entrainment. To advance fundamental understanding of the underlying physical mechanisms, a hydrodynamic model consisting of liquid displacing air in a rectangular channel in the absence of inertia is developed. Both shear thinning and shear thickening are considered by using Carreau-type models to describe the liquid rheology. Steady-state solutions are calculated using the Galerkin finite-element method and the critical capillary number where wetting failure occurs is identified. Shear thinning is found to postpone the onset of wetting failure whereas shear thickening is found to promote it. The underlying mechanism involves thickening/thinning of the air film as a consequence of shear thinning/thickening of the liquid and the tangential stress balance.

1National Science Foundation
9:18AM H08.00007 Enhanced speed of a falling sphere in pseudo-plastic fluid with ultrasound irradiation. MINORU IWAMURO, TOMOAKI WATAMURA, KAZUYASU SUGIYAMA, Osaka Univ — We experimentally investigated the effect of ultrasound irradiation on a falling sphere in pseudo-plastic fluid. The falling velocity is measured via an image processing technique. We performed experiments with parametrizing sphere diameter, fluid properties, ultrasound intensity, and its frequency. We found that the falling speed is enhanced by ultrasound irradiation, and the speed-up in the stronger pseudo-plastic fluid is much greater than that in the weaker one. We conclude that the speed-up ratio is relevant to the ratio of the viscosity and length scales involved in the system.

9:31AM H08.00008 The death of the fluid mechanical sewing machine: viscoelastic effect. BERNARDO PALACIOS, Universidad Nacional Autonoma de Mexico, STEPHEN W. MORRIS, University of Toronto, ROBERTO ZENIT, Brown University — In this work we study the fall of a viscous filament onto a moving surface. While the problem has been studied extensively for Newtonian fluids, referred to as the fluid-mechanical sewing machine, the effects of considering other complex fluids have not been explored to date. We replicate the setup used for Newtonian fluids, issuing a fluid filament from a nozzle from certain vertical distance from a moving substrate to observe it coil and stretch simultaneously. The fluid considered is, instead, a Boger fluid (viscoelastic but with constant viscosity). Our experiments show that, for similar conditions, the coiling instability does not appear if the fluid has sufficient elasticity. In most cases, the fluid mechanical sewing machine effect is not observed. Instead, a largely stable fluid catenary is observed. The shape of the catenary is characterized considering a Deborah number and a dimensionless height. A map of the conditions to kill the fluid mechanical sewing machine effect is presented in terms of these two dimensionless groups.

9:44AM H08.00009 Dynamics of viscoelastic films in reverse squeeze flows. BAVAND KESHAVARZ, ERICA LAI, GARETH MCKINLEY, NIELS HOLTE-ANDERSEN, MIT, MECHE/DNSE COLLABORATION — In many industrial and biological applications thin films of complex fluids act as lubricating layers between solid boundaries. Upon the separation of these boundaries, the kinematics of the flow generates large pressure gradients leading to high values of adhesive forces. We perform a detailed study on these phenomena using different relaxation times and elasticity moduli. The liquid is initially at rest in the gap between two circular discs. The discs are then separated from each other with an exponentially increasing velocity, ensuring a constant nominal value of stretch rate during the test. Adhesion force measurements show a rate-dependent peak force that scales with the elastic modulus of the liquid followed by an exponentially decaying tail. We show that with a proper scaling all of the measured peak forces for different viscoelastic liquids follow a single master curve. Coupling the simplified kinematics of this reverse squeeze flow with the viscoelastic constitutive equation leads to a simple lubrication model. We show that the predictions from the model agree well with the experimental measurements. Results from this study can shed light on the dynamics of liquid adhesion in complex fluids.

9:57AM H08.00010 A novel non-linear one-dimensional model for viscoelastic lubricants. LUCA BIANCOFIORE, HUMAYUN AHMED, Bilkent University — Lubrication is essential to improve the performance of sliding surfaces. Power transmission in mechanical and biological systems relies on proper lubrication to minimize wear and energy losses. However, most practical applications involve conditions that cause or require the lubricant to exhibit viscoelastic behavior. In this study a novel 1D viscoelastic Reynolds equation is derived based on the Oldroyd-B constitutive relation. It comprises a system of five 1D equations describing the pressure, velocity and shear stress distribution in the film. The model is compared with direct numerical simulations of thin films for different geometries. The results are in good qualitative and quantitative agreement indicating the simplified model is valid within the context of lubrication theory. Firstly, the pressure presents strong variations as the lubricant elasticity becomes significant, but stagnates as the polymer relaxation time becomes slow compared to the characteristic flow time. Secondly, the net film pressure is shown to be a superposition of a Newtonian and viscoelastic component. The viscoelastic component depends on the surface geometry. Surfaces with constant slope exhibit a pressure decrease, whereas the opposite effect is observed in parabolic solids.

117M434 TUBITAK 1001 grant

10:10AM H08.00011 Effect of viscoelasticity on the stability characteristics of a drying polymer solution. GEORGE KARAPETSAS, ATHANASIOS VADARLIS, Aristotle University of Thessaloniki — We investigate the stability of an evaporating liquid film which consists of a polymeric solution with a volatile solvent. Besides solutocapillary and thermal Marangoni effects, an important factor affecting the flow, which is often neglected in the literature, is the viscoelastic character of the polymeric solutions typically encountered in practical applications. During the drying process, the concentration of the solvent continuously decreases thus rendering the non-Newtonian character of the solution increasingly important. Here, we develop a model fully accounting for the viscoelastic behaviour and dynamically varying rheological properties of the liquid film. We use a finite element formulation to solve the time-dependent problem and perform a linear stability analysis employing the quasi-steady state approximation, in the limit of slow evaporation. Our numerical results indicate that the increasingly important effect of viscoelasticity destabilizes the flow and also leads to patterns with smaller wavelengths. We discuss the mechanisms which give rise to these instabilities.

1This project has received funding from the Hellenic Foundation for Research and Innovation (HFRI) and the General Secretariat for Research and Technology (GSRT), under grant agreement No 792

10:23AM H08.00012 Flow of a Shear Thickening Micellar Fluid Past a Falling Sphere. SHUIJAN WU, HADI MOHAMMADGHOUSHKI, Florida State University, FLORIDA STATE UNIVERSITY TEAM — In this work, we present the first quantitative measurements of a dilute shear thickening micellar solution past a falling sphere. The micellar solution consists of cetyltrimethylammonium bromide and 5-methyl salicylate (CTAB/5MS) in de-ionized water and it exhibits shear thickening behavior. This CTAB/5MS micellar solution forms un-entangled rod-like micelles at equilibrium. It is found that the drag coefficient for the falling sphere is similar to that of a Newtonian fluid at a vanishingly small Reynolds number (Re = 0.03). However, falling spheres experience a significant drag reduction for conditions that correspond to 0.09 ≤ Re ≤ 9.86. Moreover, an unusually extended wake which spans over a long distance downstream of the sphere is detected by particle image velocimetry. These unusual results could be rationalized by invoking the phenomenon of flow induced structure formation. We hypothesize that strong shear and/or extensional flows around the falling sphere could trigger the aggregation of rod-like micelles into giant worm-like structures. Such worm-like micelles may induce significant sphere drag reduction and extended elastic wakes in the rear of sphere. This interpretation is consistent with the steady shear and transient extensional flow measurements.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H09 Aerodynamics: General 213 - John Farnsworth, University of Colorado Boulder
8:00AM H09.00001 Enhancement of gust using jet at trailing edge of airfoil: A novel technique. DIPENDRA GUPTA, JAYWANT H. ARAKERI, Department of Mechanical Engineering, Indian Institute of Science, Bangalore — Sudden and sharp change in flow velocity, termed as a gust, is an important parameter to study flight performance, especially of micro-air vehicles (MAVs) and aircraft. There have been several techniques to create gusts in a wind tunnel including using pitching foils at the entrance of the test section. One conventional method is using two airfoils with a certain spacing to create a gust. The model aircraft to be tested is placed downstream between the pitching foils. The main limitation with this technique is the low gust intensity (lg) that can be achieved. We propose a new method to enhance lg using jet at the trailing edge of the pitching foils. Numerical simulation shows the gust intensity, using the proposed method, to increase by 2-7 times compared to that by conventional techniques for a particular Re, jet velocity and the range of reduced frequencies considered. Moreover, the spacing of foils ensures shear-free smooth flow in the near region surrounding the model, unlike some existing methods for gust enhancement. This technique provides a simple, economical and controlled way to study gust response of MAVs and aircraft in wind tunnels.

8:13AM H09.00002 The effects of freestream turbulence on the pressure distributions and lift characteristics of airfoil. LEON LI, R. JASON HEARST, Norwegian University of Science and Technology — Airfoils employed in engineering applications are often subjected to significant flow variations which can have adverse impacts on their aerodynamic performance and structural stability, for example a wind turbine operating in an atmospheric boundary layer or an aircraft in flight. The impact of these incoming (or freestream) variations is presently not well understood. To gain a better understanding, one must explore a wide parameter space of freestream parameters. This study uses 9 different passive grid configurations to alter the freestream turbulence (FST) in a wind tunnel in order to measure the impact of FST on the pressure distribution and lift characteristics of a NREL S826 reference airfoil. The FST intensity ranges from 0.5% to 5.4%, and the chord-normalized integral length scale varies between 0.07 and 0.24. The chord Reynolds number (Re) was varied between 200-400k. Preliminary results show that an increase in turbulence intensity increases both the maximum lift coefficient and the lift slope. The latter observation contrasts with some results reported in the literature with a smaller FST parameter space number (Re) was varied between 200-400k. Preliminary results show that an increase in turbulence intensity increases both the maximum lift coefficient and the lift slope. The latter observation contrasts with some results reported in the literature with a smaller FST parameter space number (Re) was varied between 200-400k. Preliminary results show that an increase in turbulence intensity increases both the maximum lift coefficient and the lift slope. The latter observation contrasts with some results reported in the literature with a smaller FST parameter space.

8:26AM H09.00003 Collaborative Experiments and Simulations of an Unsteady Free-Jet Wind Tunnel for the Study of Gust Interactions. JOHN FARNSWORTH, KENNETH JANSEN, DASHA GLOUTAK, MARK BLANCO, University of Colorado Boulder — An unsteady wind tunnel facility was recently developed at the University of Colorado by incorporating a set of counter-rotating louver vanes at the inlet to the low-speed, open-return wind tunnel. The facility was designed such that the wind tunnel can be reconfigured for testing in both a standard closed test-section configuration and an open test-section or free jet configuration. In both configurations, the wind tunnel can operate with maximum test section velocities up to 30 m/s and an unsteady peak to peak amplitude on the order of 50% of the maximum. High fidelity computational fluid dynamics simulations of the unsteady wind tunnel facility were performed and are validated against experimental measurements. Operating the facility in the closed configuration produces a velocity perturbation that propagates through the test-section nearly instantaneously and thus can be assumed to be purely unsteady. In contrast, the open test-section or free jet configuration produces a velocity perturbation that propagates near the mean convective speed of the flow through the excitation of vortical structures in the surrounding shear layer.

8:39AM H09.00004 Computational Fluid Dynamic Simulations of a Finite NACA 0015 Wing in an Unsteady Flow. MARK BLANCO, DASHA GLOUTAK, JOHN FARNSWORTH, KENNETH JANSEN, University of Colorado Boulder — Computational fluid dynamic simulations were performed for a finite span NACA 0015 rectangular wing section subjected to an unsteady surging flow in a simulated, open test-section or free jet wind tunnel. The full wing and wind tunnel facility was completely modeled and simulated for accurate comparison to collaborative experimental investigations being performed in parallel. At moderate to high angles of attack the flow over the wing section is observed to undergo periodic, three-dimensional separation and reattachment as the surging flow decelerates and then accelerates, respectively. A detailed analysis of the three-dimensional flow field behavior is discussed focusing on the time varying response of the flow field, surface pressure distribution, and wall shear stress.

8:52AM H09.00005 Experimental Measurements of a Finite NACA 0015 Wing in an Unsteady Flow as Compared to Theory. DASHA GLOUTAK, EMANUELE COSTANTINO, MARK BLANCO, KENNETH JANSEN, JOHN FARNSWORTH, University of Colorado Boulder — Force, moment and velocity measurements of a semi-span, NACA 0015 rectangular wing subjected to unsteady flow are compared to classical surging airfoil theory. Unsteady streamwise flow, generated by a louver system at the wind tunnel inlet, consisted of maximum velocity amplitudes of 40% at frequencies up to 3Hz, with mean chord Reynolds Number below 150,000. These sinusoidal velocity gusts were imposed on the wing in the closed test section and free jet wind tunnel configurations. In the former, velocity changes occurred instantaneously over the entire chord length of the wing, and in the latter the velocity changes propagated at the gust’s convective speed. Mini-sweep and unsteady lift coefficient responses were compared to Isaacs’ unsteady airfoil theory, which details the lift behavior of an infinite airfoil at constant angle of attack with a variable streamwise velocity.

9:05AM H09.00006 Boundary Layer Characterization of a NACA-0012 Airfoil Plunging in Uniform-Shear Flow using 1c-MTV. MITCHELL ALBRECHT, AHMED NAGUIB, MANOOCEHRO KOOCHESFAHANI, Michigan State University — Non-uniform approach flows can occur in environments through which aircraft navigate, such as the air wake of an aircraft carrier or aircraft formations. However, few studies have investigated the effects of viscous shear approach flow on airfoils. Our previous work shows that, in a Galilean reference frame, the lift coefficient on a NACA-0012 airfoil plunging in uniform-shear flow is greater than that of a stationary airfoil under the same flow conditions at near-stall positive angles of attack. To elaborate on this deviation from quasi-steady conditions, the current work examines the difference in lift behavior by characterizing the boundary layer around the plunging airfoil in uniform-shear flow. Single-component molecular tagging velocimetry is used to measure the streamwise velocity component of the flow on the suction side of the airfoil surface. Various boundary layer characteristics will be presented for the plunging airfoil as it traverses across the shear layer, and compared with the stationary airfoil at the same cross-stream position, flow conditions, and effective angles of attack.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0311. Computer resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility were used.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0311. Computer resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility were used.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0311. Computer resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility were used.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0311. Computer resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility were used.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-18-1-0311. Computer resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility were used.
9:18AM H09.00007 Finite Wing Hydrodynamic Forces during Water-to-Air Interface Transition. WARREN WEISLER, RAJMOHAN WAGHELA, DR. KENNETH GRANLUND, DR. MATTHEW BRYANT, North Carolina State University — Wings are used for numerous applications in both air and water and their lift generation in either domain is well understood. However, the lift generated by a wing when it is transitioning out of water and into air has not been quantified to date. This experimental study aims to examine the lift generated by a wing as it is translated through the water-to-air interface. Studies on egress velocities of 0.2 - 0.75 m/s were conducted to examine the effect of velocity on lift generation over a range of angles of attack from 0-10 degrees. The lift generated by a rectangular NACA 0015 wing with a chord of 10 cm and an aspect ratio of 4 was dependent on velocity. At the "slower" velocities there is a spike in lift when the wing leading edge nears the water surface, sometimes more than double the steady values. As the speed increases, the spike in lift disappears and lift decreases before the leading edge even reaches the surface. The results from the angle of attack testing show that the transition profile appears to scale with angle of attack.

9:31AM H09.00008 Three Dimensional Simulation of Dynamic Stall on a Dynamically Pitching Airfoil at Re = 12,000. HARRY WERNER IV, DOUGLAS BOHL, BRIAN HELENBROOK, CHUNLEI LIANG, Clarkson University — Dynamic stall (DS) is a flow separation phenomenon affecting airfoils that experience dynamic changes in Angle of Attack (AOA) beyond the airfoil’s static stall angle. Passive control of DS could extend the lifespan and efficiency of rotors, wind turbine blades, and other fixed and rotating geometric aerostructures. The development of effective methodologies for the passive control of dynamic stall requires an in-depth understanding of the fundamental flow physics governing DS phenomena. This work examines the fundamental flow phenomena of dynamic stall through Large Eddy Simulation (LES) of a dynamically pitching NACA 0012 airfoil at low Reynolds Numbers (Re = 12,000) and a constant nondimensional pitch rate of $\Omega^* = 0.1$. The simulation is a pitch and hold scenario, whereby the airfoil is rotated from 0 to 50$^\circ$AOA at a non-dimensional pitch rate of 0.1. The airfoil is held static at 50$^\circ$AOA for an additional 10 chord times to observe vortex formation and convection. Results are compared to experimental measurements.

9:44AM H09.00009 Unsteady spectra of velocity field for a canonical dynamic stall process. ROHIT GUPTA, University of Illinois at Urbana-Champaign, SABRINA HENNE, KAREN MULLENBERS, cole Polytechnique Fédérale de Lausanne, PHILLIP ANSELL, University of Illinois at Urbana-Champaign — Dynamic stall is a multiscale phenomenon that is commonly associated with airfoils undergoing rapid maneuvers. The knowledge of the dominant modes of instability in the flow field is critical for the design of effective actuation strategies for the control of dynamic stall. The objective of the present investigation was to extract the time-dependent spectral content of the velocity field about a NACA 0012 airfoil model subjected to a linear pitch-up maneuver in a water tunnel at a chord-based Reynolds number of 10^5. This objective was achieved through a novel strategy involving a combination of empirical mode decomposition and the Riesz transformations. The velocity field was acquired using high-fidelity, time-resolved particle image velocimetry. The acquired velocity measurements were processed through the empirical mode decomposition algorithm to obtain a set of oscillatory modes, known as the intrinsic mode functions, and an underlying trend, termed as the residue. The dominant amplitudes and frequencies were extracted through a direct application of the Riesz transform. The resulting velocity field spectra were then analyzed at several critical stages of the dynamic stall process.

1This study is funded by the Air Force Office of Scientific Research as part of the International Student Exchange Program (ISEP) under Agreement # FA9550-15-1-0277 in the Unsteady Aerodynamics and Turbulent Flows portfolio, monitored by Dr. Gregg Abate.

9:57AM H09.00010 Evaluation of the unsteady aerodynamic forces of an oscillating airfoil undergoing dynamic stall using impulse theory. FIRAS SIALA, JAMES LIBURDY, Oregon State University — The unsteady forces generated by a heaving and pitching airfoil are evaluated from velocity fields using the finite-domain vortex impulse theory. Time-resolved velocity fields are obtained experimentally using two-component particle image velocimetry measurements at reduced frequencies of $k = f c/U = 0.060.16$ (where $f$ is oscillation frequency, $c$ is chord length and $U$ is free stream velocity) with heaving and pitching amplitudes fixed at $h_0 = 0.6c$ and $\theta_0 = 75^\circ$, respectively. The concept of moment-arm dilemma associated with the impulse equation is revisited to shed light on its physical impact on the calculated forces. It is shown that by selecting an objectively defined origin of the moment-arm, the finite-domain impulse force formulation can be greatly reduced to two terms that have a clear physical interpretation: (i) the time rate of change of the impulse of vortical structures within the control volume and (ii) the Kutta-Joukowski force that indirectly captures the contributions of vortical structures outside of the control volume. Furthermore, it is shown that for the reduced-form of the impulse equation to be valid, a critical distance of 0.85c or greater from the airfoil trailing edge to the downstream control volume boundary is required.

1NSF

10:10AM H09.00011 Investigation of compressibility effects on a plunging airfoil under dynamic stall conditions. RENATO FUZARO MIOTTO, University of Campinas, DATTA GAITONDE, The Ohio State University, WILLIAM ROBERTO WOLF, University of Campinas, MIGUEL VISBAL, Air Force Research Laboratory — Large-eddy simulations are performed to study the compressibility effects on an airfoil under deep dynamic stall condition. In the present work, an SD7003 airfoil in plunging motion is considered at a chord Reynolds number of $Re = 60,000$ and freestream Mach numbers $M = 0.1$ and 0.4. These conditions aim to meet the current renewed interest in low- and moderate-Reynolds number unsteady aerodynamics, which finds applications in the design of small unmanned air vehicles and micro air vehicles. The current numerical methodology has already been validated for $M = 0.1$ and is herein extended for other flow regimes. In addition, modal decomposition techniques are also employed to analyze the complex off-surface flow structures associated with the dynamic stall, and to assess its onset mechanism for different compressible scenarios.

1The authors acknowledge FAPESP for the grant No. 2019/02353-6

10:23AM H09.00012 Blunt nose airfoils for improved stall performance and transonic aerodynamics. YUXIN ZHANG, WSU, MATTHEW KRALJIC, ZVI RUSAK, Rensselaer Polytechnic Institute — We numerically study the aerodynamic performance of blunt airfoils in terms of both the delay of critical Mach number for transonic shock waves to higher values and the increase of stall angle of attack at low subsonic speeds. The approach is motivated by the optimal critical airfoils developed by Schwedemman et al. (ZAMP 1993) and the recent results on the delay of stall by Kraljic & Rusak (PRFluids 2019). The nose of the airfoils is given by a canonical shape $y = (ax)^{1/\delta}$, where $\delta \geq 2$. The effect of changing the airfoils nose parameter $a$, thickness ratio $\delta$, and location of maximum thickness $x_1/c$ from leading edge on the wave drag and stall angle of attack are determined. Increasing $a$ and decreasing $x_1/c$ at a fixed $\delta$ improves lift generation and improves performance at subsonic and transonic speeds for application in wings of transport jet airplanes, rotors of helicopters, and rotating turbines.
8:00AM H10.00001 Linear Reduced-order Model based on Particle-image-velocimetry Data of Flow Field around Airfoil Controlled by Plasma Actuator. 1 KOKI NANKAI, KENTO SUZUKI, ATSUSHI KOMURO, Tohoku University, TAKU NONOMURA, Tohoku University, Presto, JST, KEISUKE ASAI, Tohoku University — A linear reduced-order model of flow fields around an airfoil controlled by a dielectric-barrier-discharge plasma actuator (DBDPA) is constructed based on particle image velocimetry (PIV) data. Velocity field data around a NACA0015 airfoil with random input by the DBDPA at the chord Reynolds number of 64,000 were acquired using PIV in a wind tunnel test. Subsequently, lower-dimensional description of the data was obtained by proper orthogonal decomposition (POD). The coefficient matrix of the linear model was computed using the least-squares approximation with the POD-mode coefficients and control input data, similar to dynamic mode decomposition with correlation. The effects of the control input component of DBDPA on the low-dimensional velocity fields reconstructed by the first ten POD modes were investigated from the input matrix. It is demonstrated that the velocity fields are more sensitive to the input change than the average value of the input. This result implies that the present model can show the well-known flow control characteristic that DBDPA with intermittent actuation such as the burst mode or nanosecond-pulse-driven mode is more effective for the separation control of flow fields than continuous actuation.

1This research is partially supported by Presto, JST (JPMIPR1678).

8:13AM H10.00002 Reduced-order Modeling and Estimation for Buoyancy-driven Flow Control. SANJANA VIJAYSHANKAR, Department of Electrical and Computer Engineering, University of Minnesota, PIYUSH GROVER, Mechanical and Materials Engineering, University of Nebraska-Lincoln, SALEH NABI, Mitsubishi Electric Research Labs — We consider the problem of data-driven reduced-order modeling and state estimation of buoyancy-driven turbulent flows in the built environment. First, we investigate the efficacy of data-driven techniques such as Eigensystem Realization Algorithm (ERA) and Dynamic Mode Decomposition (DMD) for systems described by Boussinesq equations. The resulting reduced-order models are suitable for real-time control applications. We employ these reduced-order models to construct reduced-order observers for systems operating in forced and mixed convection regimes. Exhaustive numerical simulations (both DNS and RANS) in the context of energy efficient buildings and internal flows are provided to validate the accuracy and computational benefits of this approach.

8:26AM H10.00003 Machine learning of sequential data for non-intrusive reduced-order models. ROMIT MAULIK, Argonne National Laboratory, ARVIND MOHAN, Los Alamos National Laboratory, SANDEEP MADIREDDY, BETHANY LUSCH, PRASANNA BALAPRAKASH, Argonne National Laboratory, DANIEL LIVESCU, Los Alamos National Laboratory — We study implementations of machine learning strategies for time-series data for non-intrusive reduced-order models of non-linear partial differential equations. Our reduced space is obtained with an $L_2$-optimal proper orthogonal decomposition (POD) with subsequent truncation. We then study the performance of these techniques for systems that require closure due to insufficient resolution of all of the energy in the system. Accurate non-linear dynamics in POD space are learned through recurrent neural networks and neural ordinary differential equations which utilize history, analogous to the Mori-Zwanzig formalism, to retain the effects of the unresolved modes. We also detail the use of attention to maintain the precision of learning for long-term prediction horizons and conclude by discussing distributed hyperparameter search strategies using asynchronous model-based Bayesian optimization.

8:39AM H10.00004 A Simple POD-Galerkin Model Based on Computational or Experimental Data of Flows with Moving Boundaries. MINGJUN WEI, BOLUN XU, HAOTIAN GAO, Kansas State University, JOHN HRYNUK, Army Research Lab — POD-Galerkin projection has been popular as a systematical approach for model order reduction of a complex dynamic system such as a fluid flow described by Navier-Stokes equations. However, the classical POD-Galerkin projection is derived only in a fixed domain, which limits its application on many fluid-structure interaction problems featuring moving boundaries or morphing domains. We have developed a simple modification to allow an easy implementation accounting for structural effects to extend the application of POD-Galerkin projection to a broad range of flows with moving boundaries. Recently, the modified approach was further improved to achieve better accuracy near the moving solid boundaries. The improved approach has been applied not only to direct numerical simulation data but also to experimental data. The experimental data includes the PIV measurement of the flow field for a rotating ellipse with incoming flow in a closed-loop wind tunnel.

8:52AM H10.00005 Flow Field Reconstruction and Filtering Using Spectral Proper Orthogonal Decomposition. AKHIL NEKKANTI, OLIVER SCHMIDT, University of California San Diego — The spectral variant of proper orthogonal decomposition (SPOD) decomposes a flow field into orthogonal modes that evolve coherently in both space and time, and that are optimally ranked by their energy. Just like in the case of standard proper orthogonal decomposition (POD), SPOD permits the reconstruction of the data from the modes and their expansion coefficients, and benefits from the optimality of the expansion. In this contribution, we show how the fact that SPOD is conducted in the frequency domain can be leveraged to achieve a number of goals. In particular, we use truncated series reconstructions and frequency-dependent scaling to facilitate low-rank approximations, band-pass filtering, pre-whitening and de-noising of experimental data. It is demonstrated, for example, that even a rank-1 SPOD approximation, which retains only one mode per frequency, is capable of capturing the significant dynamics of a fully turbulent flow field. Finally, we show how this iterative procedure can be employed for gappy data reconstruction. Two test cases are considered: large eddy simulation data of a turbulent jet and particle image velocimetry fields of the turbulent wake behind a flat plate at high angle of attack.

9:05AM H10.00006 Towards reduced-order SPOD-Galerkin models for turbulent flows. TIANYI CHU, OLIVER T. SCHMIDT, Department of Mechanical and Aerospace Engineering, Jacobs School of Engineering, UCSD, 9500 Gilman Drive, La Jolla, CA 92039-0411, USA — We explore the use of spectral proper orthogonal decomposition (SPOD) to construct reduced-order Galerkin models of turbulent flows under a linear time-invariant approximation. The motivation behind this particular modeling approach is the theoretical correspondence between the empirical SPOD technique and operator-based resolvent analysis, which considers optimal responses to a stochastically forced linear system. For the example of a Mach 0.9 turbulent jet, a recent study found a surprising agreement between SPOD modes computed from large-eddy simulation data and mean flow-based resolvent analysis. The same data is used in this work. Since the SPOD modes are orthogonal in a space-time inner product, the time-domain Galerkin model requires an oblique projection of the data onto the non-orthogonal modal basis. The resulting reduced-order model for the expansion coefficients is advanced in time by the linearized Navier-Stokes operator, and closed-loop control techniques, such as minimal-energy feedback control, are employed to calibrate the reduced-order model. To offset the difference between the linear approximation and the true, non-linear solution, we further incorporate the forcing statistics, inferred from applying the discrete linear operator to the data, into the model.
9:18AM H10.00007 Fast Greedy Optimization of Sensor Selection in Measurement with Correlated Noise1, KEIGO YAMADA, YUJI SAITO, Tohoku University, TAKU NONOMURA, Tohoku University, Presto, JST, KEISUKE ASAI, TOHOKU UNIVERSITY, Tsuchiura, — Efficiently leveraging limited sensor data in reduced-order models (ROMs) is key to enabling real-time control of a range of fluid flows. Two primary challenges to achieving this goal are: i) while the ROM evolves on a low-dimensional space, sensor data is typically related to the ROM via an intermediate step that involves the high-dimensional fluid state, ii) although a physical interpretation of the ROM state space may be derived, it is rarely obvious how to directly relate physical measurements to this low-dimensional representation. To address both challenges, we propose a flow-field estimation methodology where the sensor data is directly mapped to the ROM state space without involving the high-dimensional flow state. The flow-field can be efficiently recovered via the ROM approximation, if desired. We employ a neural-network architecture that learns the nonlinear mapping between the sensors and space. We emphasize that the proposed estimation framework is agnostic to the ROM employed, and can therefore be incorporated into ROMs derived by Galerkin projection, regression, etc. Our methodology is demonstrated on problems involving parametric 1D diffusion and 2D flow over an airfoil.

1This research is partially supported by Presto, JST (JPMJPR1678).

9:31AM H10.00008 Integrating sensor data into reduced-order models with deep learning, NIRMAL JAYAPRASAD NAIR, ANDRES GOZA, University of Illinois at Urbana-Champaign — Efficiently leveraging limited sensor data in reduced-order models (ROMs) is key to enabling real-time control of a range of fluid flows. Two primary challenges to achieving this goal are: i) while the ROM evolves on a low-dimensional space, sensor data is typically related to the ROM via an intermediate step that involves the high-dimensional fluid state, ii) although a physical interpretation of the ROM state space may be derived, it is rarely obvious how to directly relate physical measurements to this low-dimensional representation. To address both challenges, we propose a flow-field estimation methodology where the sensor data is directly mapped to the ROM state space without involving the high-dimensional flow state. The flow-field can be efficiently recovered via the ROM approximation, if desired. We employ a neural-network architecture that learns the nonlinear mapping between the sensors and state space. We emphasize that the proposed estimation framework is agnostic to the ROM employed, and can therefore be incorporated into ROMs derived by Galerkin projection, regression, etc. Our methodology is demonstrated on problems involving parametric 1D diffusion and 2D flow over an airfoil.

9:44AM H10.00009 Determinant based Fast Greedy Optimization on Sparse Sensor Selection1, YUJI SAITO, KEIGO YAMADA, Tohoku University, TAKU NONOMURA, Tohoku University, Presto, JST, KEISUKE ASAI, Tohoku University, DAISUKE TSUBAKINO, YASUO SASAKI, Nagoya University — The problem of optimally placing sensors to reduce calculation cost and reconstruction error arises naturally in scientific experiments. It is especially difficult to speedily select optimal sensors when the number of sensors is larger than POD modes. In this study, the authors have developed and proposed an extended determinant-based greedy algorithm based on a QR discrete empirical interpolation method (QDEIM) for the optimal sensor placement problem. The key point of this idea is that optimal sensors are obtained by the QDEIM method until the number of sensors is equal to POD modes. After that, new sensors are calculated by applying both of the determinant formula and matrix inversion lemma. We demonstrate the effectiveness of this algorithm on datasets related to climate science, and compare all calculation results; random sensors, convex approximation and a previously proposed QR algorithm in addition to the determinant-based greedy algorithm. We show that the calculation time of the proposed extended determinant-based greedy algorithm is faster than that of other methods with almost the same level of reconstruction error.

1This research is partially supported by Presto, JST (JPMJPR1678).

9:57AM H10.00010 Network-based identification of influential structures to modify turbulent flows1, MURALIKRISHNAN GOPALAKRISHNAN MEENA, KUNIHKO TAIRA, University of California, Los Angeles — The nonlinear interactions among vortical structures in turbulence make their characterization and control a challenge. We use network theory to formulate and characterize the web of interactions among vortical elements in two- and three-dimensional decaying isotropic turbulence. The nodes of these networks correspond to vortical elements in the flow field and the connections among them are weighted by the induced velocity. Network-based community detection algorithm is used to identify network connector (inter-community) and peripheral (intra-community) structures that resemble shear-layer and vortex-core type structures, respectively. We assess the influence of these structures to the neighboring vortical structures by performing DNS with added impulse perturbations to the identified network-based structures. We compare our findings with those from traditional forms of structure identification and observe enhanced turbulent mixing with the present approach. We discuss the implications of the present network-based technique for active flow control.

1This work was supported by the US Army Research Office (Grant number: W911NF-17-1-0118).

10:10AM H10.00011 Physics-informed Echo State Networks for the prediction of chaotic systems, LUCA MAGRI, FRANCISCO HUHN, University of Cambridge, NGUYEN ANH KHOA DOAN, Dept. of Mechanical Engineering, Technical Univ of Munich — We suggest Echo State Networks (ESN), a data science technique, to predict the evolution of chaotic dynamical systems, namely those of high-fidelity fluid dynamics simulations, LES and DNS. Data generated from high-fidelity simulations carry high computational cost and thus only small amounts are available. While this usually poses a limitation to data science techniques – unlike the traditional big data, this problem lives in the world of “small data” –, this can be balanced by leveraging physical knowledge of the system in study, that is, while ESNs can be trained purely on past observations, their performance can be improved, for example, by including a loss term that represents the system’s physics and penalizes non-physical predictions in the training phase. We explore the characteristics and performance of physics-informed ESN models, from a nonlinear dynamics point-of-view, in reproducing chaotic dynamical systems. Finally, we look into potential applications to fluid dynamics problems, such as prediction of extreme events or sensitivities of time-averaged cost functionals.

10:23AM H10.00012 Dimensionality Reduction and Reduced Order Modeling for Traveling Wave Physics1, ARIANA MENDIBLE, ALEKSANDER ARAVINK, University of Washington, WES LOWRIE, STEVEN BRUNTON, J. NATHAN KUTZ, University of Washington — Large scale spatiotemporal data are ubiquitous across many fields of science and engineering, especially in fluids. Standard dimensionality reduction techniques based on the singular value decomposition (SVD) often fail to provide a compact representation of traveling waves because the SVD is inherently a space-time separation of variables. This necessitates a data-driven method to decompose and reduce spatiotemporal systems with multiple traveling waves. In this work, we investigate alternative approaches to dimensionality reduction that are designed specifically to separate and represent interpretable traveling wave structures in a low-dimensional form. Using a shifted SVD, we formulate an unsupervised, optimization-based framework for identifying parsimonious wave speed models from many candidate models. We demonstrate our method on example systems which pose challenges of non-periodicity, nonlinearity, and changing wave speed.

1HDTTRA1-18-1-0038
8:00AM H11.00001 An Experimental Investigation of Surfactant Effects on the Wave Characteristics of Annular Flows\textsuperscript{1}, ANDRIUS PATAPAS, VICTOR VOULGAROPOULOS, VALERIA GARBIN, RONNY PINI, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London, KARL ANDERSON, Shell — While a wide scope of research has been performed in the field of multiphase flows, the study of surface active agents in gas-liquid annular flows has lagged despite their ability in considerably improving heat and mass transfer rates. More detailed insight is still needed to promote quantitative interpretations of the surfactant-induced effects on the wave characteristics and entrainment properties of these thin films. In this work, we study water-air annular pipe flows in both the presence and absence of a water-soluble fluorescent surfactant. The liquid Reynolds numbers, based on the film properties, range between 400 to 1500. We perform structured planar laser-induced fluorescence (S-PLIF) measurements to accurately obtain film-thickness measurements and reveal the temporal characteristics of the waves. We further explore the differences on the entrainment rates and size of the bubbles in the liquid films for both cases, by visualising the flow from two observation angles. Surfactant-tracking diagnostic methods are also currently being developed.

\textsuperscript{1}Funding from Shell and the Transient Multiphase Flows Consortium is gratefully acknowledged

8:13AM H11.00002 Laser Cantilever Anemometry for highly resolved velocity measurements in fluids, JAROSLAW PUCZYLOWSKI, University of Oldenburg, MEASUREMENT SCIENCE ENTERPRISE MSE COLLABORATION — Laser Cantilever Anemometry is a new and high-resolution measurement method for determining the velocity vector of a fluid. A micro-structured silicon cantilever serves as a drag body that is set into the flow. The forces acting on the cantilever, which are exerted by the moving fluid particles, cause the cantilever to bend or twist. With the help of the laser pointer principle, this deformation can be detected and finally recalculated into a velocity vector in two dimensions. The measuring principle is characterized by a very high temporal and spatial resolution (approx. 100m at 100kHz). In the recent past a sensor was developed and tested, which uses this measuring method. The so-called 2d-LCA (2d-Laser Cantilever Anemometer) was extensively and successfully used under different laboratory conditions. The current version of the 2d-LCA has the size of a highlighter and is operated via USB-C or Bluetooth. This compact design is extremely portable and allows the use in difficult to access areas. In addition, all hardware components of the 2d-LCA are made of Invar (Alloy36), which has a very low coefficient of thermal expansion. This avoids heat-related drift and leads to a very stable signal. At the The 72nd Annual Meeting of the American Physical Society’s Division of Fluid Dynamics (DFD), the measuring principle of the 2d-LCA will be explained in more detail and measurement results will be presented.

8:26AM H11.00003 Development of Focused Laser Differential Interferometry for Hypersonic Freestream Measurements\textsuperscript{1}, JOEL LAWSON\textsuperscript{2}, MALLORY NEET\textsuperscript{3}, JOANNA AUSTIN\textsuperscript{4}, California Institute of Technology — Focused laser differential interferometry (FLDI) is a non-invasive diagnostic capable of making localized density measurements with high temporal resolution. Its distinguishing feature for ground testing is diminished response away from the focal plane, thereby mitigating signal contributions from fluid not in the facility core flow. We first present a quantitative experimental validation of a ray-tracing scheme used to model the FLDI response. This is followed by some applications of the technique to Caltech’s hypersonic ground testing facilities: firstly, FLDI is applied to the Hypervelocity Expansion Tube (HET) to measure the freestream noise spectrum during test time, and relating this to the initial driver gas state as per the acoustic wave theory of Paull and Stalker [J. Fluid Mech., vol. 245, pp. 493-521, 1992]. Secondly, FLDI is used to track a laser-induced breakdown in the freestream of the T5 reflected shock tunnel, with the goal of measuring flow velocity and sound speed.

\textsuperscript{1}Office of Naval Research award N00014-16-1-2503
\textsuperscript{2}PhD Candidate, Graduate Aerospace Laboratories
\textsuperscript{3}PhD Candidate, Graduate Aerospace Laboratories
\textsuperscript{4}Professor of Aerospace, Graduate Aerospace Laboratories

8:39AM H11.00004 Weighted Least squares density reconstruction for Background Oriented Schlieren (BOS)\textsuperscript{1}, LALIT RAJENDRAN, JIACHENG ZHANG, SALLY BANE, PAVLOS VLACHOS, Purdue University — Background Oriented Schlieren (BOS) is an optical technique used to measured density gradients by tracking the apparent distortion of a dot pattern. The density gradient field can then be spatially integrated to calculate the density field. The Poisson solver is currently the standard for density integration, but it is sensitive to noise in the gradient field. We address this limitation and improve the overall accuracy of the density integration process by employing a weighted least-squares (WLS) method. WLS performs the 2D integration of a gradient field by solving an over-determined system of equations. Weights are assigned to the grid points based on errors/uncertainties in the density gradient field to ensure that a less reliable measurement point is given less weight on the integration procedure. This method has been shown to increase the robustness of the integration in velocity-based pressure estimation, and in this study we will assess its suitability for BOS. We will assess the calculation of weights based on two methods: (1) based on the displacement uncertainty and (2) using a curl-free constraint on the density gradient field, as the underlying density field is a scalar. The contribution is an improved integration method for density estimation from BOS.

\textsuperscript{1}This material is based upon work supported by the U.S. Department of Energy, Office of Science, Office of Fusion Energy Sciences under Award Number DE-SC0018156.
A quantitative analysis of the chemical evolution of an iodine plume using optical filtering, imaging spectroscopy, and schlieren imaging. SARAGORIO, ALEXANDRA RIVERA, MICHAEL HARGATHER, New Mexico Tech — The chemical evolution of an iodine plume was quantitatively analyzed using a dual high-speed camera imaging system and verified using imaging spectroscopy. The dual high-speed camera imaging system consisted of a parallel light lens schlieren system with a beamsplitter located after the knife-edge to send the light into two Phorion Mini AX cameras. Each camera imaged through an optical notch filter, one at 520 nm and one at 650 nm. The 520 nm and 650 nm filters correspond to the maximum absorption wavelength and zero absorption wavelength of the iodine absorption spectra, respectively. A turbulent plume, consisting of vaporized iodine and a carrier gas, was imaged in the dual camera system. The resulting image sets were processed to relate differences in pixel intensity to light absorption intensity of the developing plume. The plumes iodine concentration was then derived from the changes in absorption intensity. A validation system, using a Horiba MicroHR imaging spectrometer and a Phorion SA-X2 camera, measured the iodine plumes absorption spectra directly and verified the accuracy of the optical filtering technique. Results show the ability to perform simultaneous refractive imaging and species identification in a turbulent flow field.

**9:05AM H11.00006** Measurement of high frequency temperature fluctuations in high-pressure premixed combustion using laser Rayleigh scattering, HAN JUNE PARK, SANGEUN BAE, HYUNGROK DO, WONTAE HWANG, Seoul National University — In gas turbines, the high temperature and pressure combustion environment makes it difficult to measure physical properties such as flame characteristics. This measurement issue is also crucial when assessing combustion instability. Thermoacoustic combustion instability is known to occur when heat release and acoustic oscillations become in phase with each other. Due to the nature of heat release oscillation, modeling and measurement are difficult. To measure heat release oscillations, temperature fluctuation measurements are necessary. However, previous research in high temperature and pressure environments has been limited. In this study, we measured high frequency temperature fluctuations in high-pressure premixed combustion through density fluctuations, using non-intrusive laser Rayleigh scattering. The photon counting method was applied to quantify low intensity Rayleigh scattering. In addition, a high-speed camera was used to correlate the flame shape and combustion instability as pressure increases.

**9:18AM H11.00007** Planar spray visualization processing techniques and considerations, KYLE M. BADE, RUDOLF J. SCHICK, Spraying Systems Co., SPRAY ANALYSIS AND RESEARCH SERVICES TEAM — Techniques to visualize and characterize spray distributions include mechanical and optical methods. Planar laser illumination presents a non-intrusive and direct method to visualize and quantify the size, shape, and distribution of a spray across a known cross-section. The nuances of processing and interpreting the light intensity from polydisperse sprays can have a significant effect on the resulting relative distribution of the scattered light. Mie scattered light is proportional to the surface area of the droplet, while laser induced fluorescence (LIF) emits an intensity proportional to the spray volume. As a result, processing of instantaneous droplet fields using both techniques may recover an effective Sauter Mean Diameter (D32), this process is explained and demonstrated. It is well understood that collected scattered light intensity is proportional to the number of droplets as well as the size of those droplets, with increasing numbers and diameter deliver a higher intensity. Additionally, the scattering angle and position of a droplet relative to the illumination source and collection device (camera) effect the perceived local concentration. The effects of these, and other details, are demonstrated and explained in the context of the Spraying Systems Co. SprayScan™ mPT instrument as well as for general experimental setups.

**9:31AM H11.00008** 2D X-ray Radiography and 3D Computed Tomography of a Spray, DANYU LI, THEODORE J. HEINDEL, Iowa State University, EXPERIMENTAL MULTIPHASE FLOW LAB TEAM — X-ray computed tomography can provide a comprehensive three-dimensional time-average view of a spray with clear internal details. Compared with general X-ray radiography, however, it is time-consuming and the 3D reconstructions require specialized imaging tools. For axisymmetric sprays, it is possible to create a full density distribution from the 2D X-ray radiographs. In this research, broadband tube source X-rays were used to take 2D radiographs of a spray from a two-fluid coaxial nozzle. To enhance image contrast, 20% potassium iodide by mass was added to the water. Based on the circular symmetry of the spray, an Abel inversion was used to analyze the time-average radial density distribution of the spray. X-ray computed tomography was also used to quantify the internal structure and the radial density distribution of the identical spray. The results of the two were compared and showed good agreement as long as the spray is axisymmetric.

**9:44AM H11.00009** A Near-field Comparison of Flows at Similar Momentum Ratios in a Co-Flow Airblast Atomizer, THOMAS J. BURTNETT, TIMOTHY B. MORGAN, DANYU LI, JULIE K. BOTHELL, THEODORE J. HEINDEL, Iowa State University, ALBERTO ALISEDA, NATHANIAEL MACHICOANE, University of Washington, ALAN L. KASTENGREN, Argonne National Laboratory, EXPERIMENTAL MULTIPHASE FLOW LAB TEAM — Spray atomizers can be found in a wide variety of applications, from combustion systems to nasal inhalers. Capturing the dynamics of sprays and their formation in the near-field region are often challenging due to their optically dense nature. In this study, a co-flow airblast atomizer was used to disperse a stream from an inner liquid needle. Gas and liquid flow rate combinations were varied to achieve conditions with similar gas-liquid momentum ratios. The high-resolution X-ray capabilities at the Advanced Photon Source at Argonne National Laboratory were used to characterize the near-field spray formation region. Quantitative results from focused-beam X-ray measurements, such as the fluid’s equivalent path length (EPL), were determined and compared for various momentum ratios. Similar momentum ratios provided similar EPL measures in selected regions, even while the gas and liquid Reynolds numbers were varied. Regions where differences appeared for the same momentum ratio are also identified and discussed.
Turbulent environments are abundant in nature, yet they are still not completely understood. Research on particles in turbulence has focused mainly on spherical particles smaller than the viscous scale even though many organisms found in nature exceed the size of the turbulent viscous length-scale and are not spherical in shape. Sea Walnuts and other zooplankton fall into this shape and size ranges of interests. These organisms have long been considered important species in ocean ecosystems. I am interested in understanding how these organisms exchange soluble material with their environment. Using dissolving sugar particles as a proxy, I investigate the fundamental questions of how shape, surface area, and volume affect particle dissolution rates in a laboratory turbulence tank. The research thus far has shown that neither surface area nor volume alone govern the dissolution rate of dissolving particles. Determining a single governing parameter may help elucidate how organisms interact with their environment and can contribute to the design of mini robots with respect to how they sample concentration plumes in rivers, streams, oceans, and other bodies of water.

10:10AM H11.00011 Multiphase liquid entrainment measurements in high pressure piping

AMY MCCLENEY, Southwest Research Institute — A two-phase flow of natural gas and liquid heptane flowing in 3-inch and 6-inch diameter pipes pressurized up to 3,600 psig is characterized using a combination of a novel liquid sampling technique and a flow imaging technique. The results from this new measurement approach show how the phase distribution inside of the piping can be obtained as well as how the liquid entrainment fraction changes with an increase in gas and liquid velocity. In the petroleum industry, multiphase flows are encountered during natural gas production and processing. In two-phase flow scenarios, the amount and distribution of liquid in the gas stream play an important role in the selection and operation of flow measurement devices and the design of gas processing and separation equipment. The improper design of these types of equipment can negatively affect the allocation of petroleum resources. Several experimental tests have been conducted to investigate and characterize two-phase flows in pipes with low liquid levels but are based on experimental data collected at pressures less than 10 psig. This study indicates that the pressure has a significant effect in the distribution and behavior of two-phase flow mixtures, as the natural gas will dissolve in the liquid phase as the pressure increases.

10:23AM H11.00012 Optical measurements of the velocity, height and frequency of disturbance waves in saturated two-phase annular flow

THERESA B. OEHMKE, EVAN A. VARIANO, University of California, Berkeley — In two-phase annular flows, disturbance waves, i.e., large waves several times thicker than the mean thickness of the liquid film, strongly affect the heat transfer coefficient and pressure drop. In this study, an optical technique is used to simultaneously measure the instantaneous velocity and height, as well as the frequency of disturbance waves in an adiabatic two-phase annular flow of saturated R245fa. Instantaneous liquid film thickness is measured using the optical method initially proposed by Shedd and Newell (1998, https://doi.org/10.1063/1.1149232). In this method, the liquid film thickness is measured by ring patterns made by light reflected at the interface of the liquid film. A new strategy for the post-processing of the ring patterns has been developed that allows the measurement of the instantaneous velocity, height, and frequency will be presented for flows with vapor qualities between 0.63 and 0.9. Results show that disturbance waves become slower and less frequent at the high vapor qualities.

---

Monday, November 25, 2019 8:00AM - 10:36AM – Session H12 Non-Linear Dynamics: Coherent Structures II

8:00AM H12.00001 Energy cascades and the asymmetric motion of coherent structures

DOUGLAS KELLEY, JEFFREY TITHOF, University of Rochester, GERRIT HORSTMANN, University of Rochester and Helmholtz Zentrum Dresden-Rossendorf, BALACHANDRA SURI, IST Austria, HUSSAIN ALUIE, University of Rochester, MICHAEL SCHATZ, ROMAN GRIGORIEV, Georgia Tech — We present evidence for a connection between energy cascades, which transfer energy among length scales in turbulent flows, and the Lagrangian coherent structures (LCS), which describe passive scalar transport in flows. LCS come in two types, which mark regions of strongest nonlinearity in forward- and backward-time flow, respectively. But prior observations have shown a time asymmetry: in turbulent flows, and the Lagrangian coherent structures (LCS), which describe passive scalar transport in flows. LCS come in two types, which mark regions of strongest nonlinearity in forward- and backward-time flow, respectively. But prior observations have shown a time asymmetry: in real 2D flows, the Lagrangian coherent structures are highly dissipative. In this study, we find that disturbance waves become slower and less frequent at high vapor qualities.

8:13AM H12.00002 Relating 2D and 3D Lagrangian coherent structures in oceanic flows

H M ARAVIND, G. SALVADOR-VIEIRA, Northeastern University, VICKY VERMA, SUTANU SARKAR, University of California San Diego, MICHAEL ALLSHOUSE, Northeastern University — Lagrangian coherent structures (LCS) techniques reveal elliptic structures that partition the domain into minimally- and well-mixed regions (Jupiter’s Great Red Spot, Polar Vortex), and hyperbolic structures that are highly attracting/repelling material surfaces (long filaments of the Deepwater Horizon oil spill). These techniques can be applied to 3D ocean flows; however, the high computational cost of calculating 3D LCS or the lack of an accurate vertical velocity model often prevent the use of LCS techniques. In this study, we compare 2D FTLE and trajectory-clustering structures with 3D results for a high-fidelity, submesoscale simulation of an oceanic density front.

1Support provided by ONR N00014-18-1-2790
8:26AM H12.00003 Vortex boundaries as barriers to vorticity transport in two-dimensional flows\(^1\), STERGIOS KATSANOUILIS, ETH Zurich, MOHAMMAD FARAZMAND, Massachusetts Institute of Technology, MATTIA SERRA, Harvard University, GEORGE HALLER, ETH Zurich — Recent advances have revealed barriers to diffusive transport as material curves that inhibit the transport of diffusive scalars more than neighboring curves do. Extending these results, we discuss a new, fully frame-independent (objective) vortex identification method for two-dimensional flows. Our method locates vortex-core boundaries as closed material curves that inhibit the diffusion of vorticity more than other nearby material curves do. The exact solution to this calculus of variations problem provides a criterion that unites common features of empirical observations: the material and vorticity-transporting nature of observed vortex cores. We also discuss an algorithm, along with a publicly available numerical package, that enables the automatic extraction of maximally vorticity-preserving, material vortex cores from two-dimensional data sets. We conclude by demonstrating this algorithm on explicit Navier-Stokes solutions and two-dimensional turbulence simulations.

\(^1\)Turbulent Superstructures priority program, German National Science Foundation (DFG)

8:39AM H12.00004 Material barriers to the transport of momentum and vorticity\(^1\), GEORGE HALLER, ETH Zurich — Recent work has identified objective (frame-indifferent) material barriers that are the least impermeable to the diffusion of passive scalars in turbulent flows. Here we discuss the extension of these results to identify material barriers to the transport of active quantities, such as vorticity and momentum, in three-dimensional unsteady flows. Challenges in such an extension include the vectorial nature of these active quantities, as well as their dependence on the frame of reference. With these challenges addressed, we obtain a general algorithm for locating objective material barriers to active transport, which form a dynamical skeleton around possible pathways in the flow. We illustrate the results on closed-form solutions of the Navier-Stokes equations, and well as on three-dimensional numerical simulations.

\(^1\)Partially supported by the Turbulent Superstructures program of the German National Science Foundation

8:52AM H12.00005 Stochastic Lagrangian Dynamics of Vorticity in Wall-Bounded Flows: General Theory\(^2\), GREGORY EYINK, The Johns Hopkins University, AKSHAT GUPTA, Technical University of Munich, TAMER ZAKI, The Johns Hopkins University — The Lagrangian properties of vorticity for a smooth Euler solution have been extended to Navier-Stokes solutions by Constantin-Iyer (2008,2011) using a stochastic Lagrangian approach. This is best understood within the Kuz'min-Oseledeits formulation of Navier-Stokes, in terms of the “vortex-momentum” associated to a continuous distribution of infinitesimal vortex rings. This theory provides an infinite set of exact Lagrangian conservation laws for Navier-Stokes vorticity, the “stochastic Cauchy invariants”. These are preserved only backward in time, due to the irreversibility of Navier-Stokes dynamics. For wall-bounded flows, these invariants allow a complete representation of interior vorticity in terms of vorticity generated at a solid wall, as it is advected, stretched and rotated by the flow. We present a Monte Carlo method to calculate the stochastic Cauchy invariants and their statistics by solving the SDE’s for ensembles of stochastic Lagrangian particles. We test the method using a space-time database of turbulent channel-flow at \(R_e = 1000\), verifying the conservation of mean values of the stochastic Cauchy invariants. Their variances grow exponentially in time, reflecting Lagrangian chaos in the channel flow and implying large cancellations in the conserved means.

\(^2\)G.E. and A.G. supported by NSF OCE-1633124
eyink@jhu.edu

9:05AM H12.00006 Stochastic Lagrangian Dynamics of Vorticity in Wall-Bounded Flows: Turbulent Channel-Flow\(^1\), AKSHAT GUPTA, Technical University of Munich, GREGORY EYINK, TAMER ZAKI, The Johns Hopkins University — We exploit an exact stochastic Lagrangian formulation of Navier-Stokes to study vorticity dynamics in a turbulent channel-flow at \(R_e = 1000\). “Stochastic Cauchy invariants” are conserved on average backward in time along stochastic Lagrangian particle trajectories, even as individual vorticity vectors are advected, stretched and rotated. At close prior times, conservation requires delicate cancellations between vorticity contributions from particles in the interior of the flow and those which strike the wall and remain fixed there. Far back in time, interior vorticity is represented by an average over vorticity that originated entirely at the wall. As in superfluids, cross-stream transport of tangential vorticity generated at the wall is exactly related to drag. We show that the process of vortex-lifting in the buffer layer is not an abrupt lifting of discrete vortex lines but is instead distributed over 100’s of viscous times and 1000’s of wall units. Despite simple arrays of “hairpin” vortex-lines, the dynamics involves intense competition between nonlinear Lagrangian chaos that exponentially magnifies \& rotates vorticity and strong viscous destruction. We discuss the Lighthill-Morton theory of vorticity generation from this stochastic Lagrangian perspective.

\(^1\)G.E. and A.G. supported by NSF OCE-1633124
alaksat.gupta@tum.de
eyink@jhu.edu

9:18AM H12.00007 True topology of 3D unsteady flows with spheroidal invariant surfaces\(^1\), MICHEL SPEETJENS, SEBASTIAN CONTRERAS, HERMAN CLERCX, Eindhoven University of Technology — Scope is the response of Lagrangian flow topologies of 3D time-periodic flows consisting of spheroidal invariant surfaces (ISS) to perturbation. Such ISS have intra-surface Hamiltonian topologies comprising of islands and chaotic seas. Computational studies predict a response to perturbation dramatically different from the classical case of toroidal ISS: said islands and chaotic seas evolve into ‘tube-and-shell’ structures by ‘resonance-induced merger’ (RIM). This study provides conclusive experimental proof of RIM and advances the corresponding structures as the true topology of realistic flows with spheroidal ISS; the latter are singular entities that exist only for ideal conditions. Theoretical analysis reveals that RIM ensues from perturbed periodic lines via two possible scenarios: truncation of tubes by (i) manifolds of isolated periodic points emerging near elliptic lines or by (ii) segmentation of lines into elliptic and hyperbolic parts. This Furthermore demonstrates that RIM indeed accomplishes tube-shell merger by exposing the existence of ISS that smoothly extend from tubes into chaotic shells. These phenomena set the response to perturbation – and true topology – of flows with spheroidal ISS fundamentally apart from flows with toroidal ISS.

\(^1\)Consejo Nacional de Ciencia y Tecnología (CONACYT)
9:31AM H12.00008 Critical point identification in 3D velocity fields.¹, MOHAMMADREZA ZHARFA, PAUL S. KRUEGER, Southern Methodist University — Classification of flow fields involving strong vortices such as those from bluff body wakes and animal locomotion can provide important insight to their hydrodynamic behavior. Previous work has successfully classified 2D flow fields based on critical points of the velocity field and the structure of an associated weighted graph using the critical points as vertices. The present work focuses on extension of this approach to 3D flows. To this end, we have used the Gauss-Bonnet theorem to find critical points and their indices in 3D vector field, which functions similarly to the Poincare-Bendixon theorem in 2D flow fields. The approach utilizes an initial search for critical points based on local flow field direction, and the Gauss-Bonnet theorem is used to refine the location of critical points by dividing the volume integral from of the Gauss-Bonnet theorem into smaller regions. To verify this approach, we have applied this method on some flow fields generated from potential flow theory and numerical methods.

9:44AM H12.00009 Spectral reconstruction of incompressible flows and application to incomplete flow measurements, SIAVASH AMELI, SHAWN SHADDEN, UC Berkeley — A wide range of fluid flow applications are incompressible. Noise in flow measurements is the main source that violates the divergence-free condition for such flows. A variety of approaches have been proposed to filter noise and reconstruct data. Proper Orthogonal Decomposition, Dynamic Mode Decomposition, radial basis functions and smoothing kernels and spectral filtering by Fourier representation are a few examples. Yet, for many applications the necessity of an incompressible projection is important. Previously, we have presented the spectral representation of vector fields that addresses incompressibility by means of orthogonal family of solenoidal fields. The spectral representation of the fluid flow is obtained by the projection of the flow to the eigenmodes that are generated for a specific geometry and satisfy a flow boundary condition. In this talk, we discuss the optimality of the set of eigenfunctions in modal reduction and representation of incompressible flows. Also, we will present modal reconstruction of the flow to the eigenmodes that are generated for a specific geometry and satisfy a flow boundary condition. In this talk, we discuss the optimality of the set of eigenfunctions in modal reduction and representation of incompressible flows.

9:57AM H12.00010 Learning the Tangent Space of Dynamical Instabilities from Data, ANTOINE BLANCHARD, THEMISTOKLIS SAPSIS, MIT — The optimally time-dependent (OTD) modes are a set of deformable orthonormal tangent vectors that track directions of instabilities along any trajectory of a dynamical system. Traditionally, these modes are computed by a time-marching approach that involves solving multiple initial-boundary-value problems concurrently with the state equations. However, for a large class of dynamical systems, the OTD modes are known to depend “pointwise” on the state of the system on the attractor, and not on the history of the trajectory. We leverage the power of neural networks to learn this “pointwise” mapping from phase space to OTD space directly from data. The result of the learning process is a cartography of directions associated with strongest instabilities in phase space, as well as accurate estimates for the leading Lyapunov exponents.

10:10AM H12.00011 Search and Rescue at Sea Aided by Hidden Flow Structures, MATTIA SERRA, Harvard University, PRATIK SATHE, UCSB, IRINA RYPINA, ANTHONY KIRINICICH, WHOI, SHANE ROSS, Virginia Tech, PIERRE LERMUSIAUX, THOMAS PEACOCK, MIT, ARTHUR ALLEN, U.S. Coast Guard - Search and Rescue, GEORGE HALLER, E TITIN Zurich — Every year hundreds of people die at sea because of vessel and airplane accidents. Using recent mathematical results for assessing short-term material transport in unsteady flows, we uncover hidden Transient Attracting Profiles (TRAPs) in ocean-surface velocity data. Computable from a single velocity-field snapshot, TRAPs act as short-term attractors for all floating objects. We emulate SAR scenarios in three different ocean field experiments, and show that TRAPs computed from measured as well as modelled velocities attract deployed drifters and manikins emulating people fallen in water. TRAPs, which remain hidden to prior Eulerian diagnostics, thus provide critical information for hazard responses, such as SAR and oil spills, and have the potential to save life and limit environmental disasters.

10:23AM H12.00012 Turbulent Flow in the Vicinity of Retaining Walls: Conditions at the Early Stages of Local Scour Development, NASSER HEYDARI, PANAYIOTIS DIPLAS, Department of Civil and Environmental Engineering, Lehigh University, Bethlehem, PA, J. NATHAN KUTZ, Department of Applied Mathematics, University of Washington, Seattle, WA — Turbulent flow characteristics corresponding to the initial stages of local scour development are investigated in the vicinity of a retaining wall structure. Data collection is performed using a volumetric particle image velocimetry to measure the three-dimensional velocity fields. Time-averaged flow topology, turbulence statistics, and instantaneous fields are examined. Furthermore, proper orthogonal decomposition (POD) and dynamic mode decomposition (DMD) tools are applied to further understand the underlying features of the large-scale coherent structures around the obstruction. The results indicate that a horseshoe vortex (HV) system develops over the channel bank at the upstream face of the obstruction. It follows the sloping junction line towards the toe of the channel bank and then bends in the direction of the flow. It was indicated that turbulent kinetic energy (TKE) inside the HV system and bed shear stress values in the mean flow are more pronounced near the leading edge of the protrusion where the flow acceleration is the strongest. Consistently, the leading POD and DMD modes indicate that the HV captures a significant portion of the TKE content. They also confirm the aerodynamic behavior of the HV system.

¹The support of the National Cooperative Highway Research Program [grant NCHRP-HR 2436] for this study is gratefully acknowledged.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H13 Convection and Buoyancy-driven Flows: General 304 - Thomas Solano, Florida State University

8:00AM H13.00001 Cafe’ Latte: Spontaneous layer formation in laterally heated double diffusive convection: Numerical simulations¹, KAI LEONG CHONG, RUI YANG, University of Twente, ROBERT VERZICCO, University of Roma, DETLEF LOHSE, University of Twente — The experiments of N. Xue et al., Nat. Commun. 8, 160 (2017) have shown that the well-known layer formation in Cafe’ Latte is due to double diffusive convection, with the hot milk injected into the coffee cooling down to the side. Here we perform a corresponding numerical investigation of laterally cooled double diffusive convection where the fluid flow is driven by a horizontal temperature difference and stabilized by a vertical concentration difference. When the stabilization caused by the concentration field is too weak, the flow behaves like standard vertical convection in which there is a large-scale circulation throughout the domain. However, upon increasing stabilization, one observes the spontaneous formation of layers where there are several vertically-stacked circulations and every circulation forms a well-mixed layer. As time evolves, the two neighbouring layers can merge into one layer whenever there is strong enough flow that can break the interface in between. Finally, we analytically calculate the average initial height of the layers and find good agreement with our numerical data.

¹ERC advanced grant
8:26AM H13.00003 Propagating fronts through convective flow fields with cooperative and antagonistic feedback1. SAIKAT MUKHERJEE, MARK PAUL, Virginia Tech — We numerically explore propagating fronts generated by an exothermic and autocatalytic reaction where the products and reactants can vary in density. We study fronts traveling horizontally in a long, shallow, and two-dimensional layer of fluid undergoing thermal convection. In the absence of heat generation by the reaction, any variation in density between the products and reactants results in a single solutally-driven convection roll that propagates with the front. In the presence of heat release by the reaction, a hotspot at the front is generated which leads to the formation of a pair of counter-rotating convection rolls that travel with the front. When the products are less dense than the reactants, the thermal and the solutal effects are cooperative. When the products are more dense than the reactants, the thermal and solutal effects are antagonistic. We study fronts with cooperative and antagonistic feedback that travel through counter-rotating convection rolls generated by Rayleigh-Bénard convection. In the presence of convection, the front and fluid dynamics exhibit oscillatory dynamics. We quantify the fluid dynamics, front velocity, and reaction length scale over a wide range of solutal and thermal driving.

1Partial support by DARPA Grant No. HR0011-16-2-0033

8:39AM H13.00004 Conjugate Thermal Boundaries Effect On Natural Convection Flow Structures In Enclosures. TOMAS SOLANO, JUAN ORDONDEZ, KOUROSH SHOOLE, Florida State University — Buoyancy-induced flow for a heated sphere in a square enclosure coupled with external forced convection cooling of the enclosure walls is investigated numerically using an immersed boundary method and conjugate thermal boundary conditions. The external forced convection cooling, on the enclosure boundaries, is seen to significantly modify the internal convectionary flows, i.e. flow and thermal stratification depending on Rayleigh number and aspect ratio. The relation between the external Reynolds number and the internal Rayleigh number is reported in terms of streamlines, isotherms, local and average Nusselt numbers. Asymmetric and reverse flow is observed as the relation between external Reynolds and internal Rayleigh is changed. The coupling between the two modes of convection is presented as a means to control the convection pattern and vertical structures in an enclosure with heated components.

8:52AM H13.00005 Thermal convection in a rotating liquid sphere with radial gravity as a direct function of the radius.. VICTOR HUITRON, RUBEN AVILA, Facultad de Ingeniería, Universidad Nacional Autónoma de México, México City, Mexico — The thermal convection of the terrestrial planets without solid inner core, has been subject of research in the last decades. Some theories claim that at the beginning of the formation of the planets, the solid inner core was absent. This research is aimed to analyze the convective patterns and the heat transfer rate in a rotating liquid sphere by solving the non-steady, three-dimensional Navier-Stokes equations. In order to avoid the singularity at the center of the sphere, the set of equations for an incompressible fluid are formulated in a Cartesian coordinate system and solved by using the mesh-based h/p Spectral Element method. The thermal convection is driven by both a uniform internal energy source and a radial gravitational field that is directly proportional to the radius of the sphere. The effect of increasing the value of the Rayleigh Ra number and the Taylor Ta number, on the convective patterns and on the velocity, temperature and vorticity fields is presented. The local and average Nusselt numbers at the surface of the whole sphere are evaluated together with the temperature, vorticity and pressure fields. The obtained results are successfully compared with numerical solutions previously published in the literature.

9:05AM H13.00006 Diagnosis of turbulence radiation interactions in turbulent premixed flames under high-pressure and high-Ka conditions. BIFEN WU, XINYU ZHAO1, University of Connecticut — Predicting thermal radiation is crucial for prediction of wall heat transfer for aeronautical combustors. Intense turbulent fluctuations result in strong fluctuations of temperature and composition, which lead to highly nonlinear behavior in the radiative source terms that is conventionally referred to as to turbulence-radiation interactions (TRI). An a priori study of the TRI is performed using datasets from Direct Numerical Simulation (DNS) of strongly turbulent premixed n-dodecane/air flames with three Karlovitz numbers (Ka) ranging from 100 to 10000, under pressurized conditions that are relevant for engine operations. Here, a line-by-line database derived from HITEMP2010 and HITRAN2018 is employed to calculate the radiative properties of radiative species. The database is extrapolated to high-pressure conditions using the Voigt profile. The influence of various interactions in the radiative emission source term is investigated and their impact is evaluated in the context of large eddy simulations (LES) through explicit filtering with different filter sizes. Sub-grid scale TRI models of different fidelities, such as no-TRI, partially-TRI, are assessed. Different levels of soot volume fractions are also artificially introduced to investigate the influence of soot on TRI.

1Corresponding author

9:18AM H13.00007 Weakly nonlinear stability analysis on a chemotaxis system with deformed free surface. SYMPHONY CHAKRABORTY, TONY WEN-HANN SHEU, National Taiwan University, No. 1, Sec. 4, Roosevelt Road, Taipei 10617, Taiwan (R.O.C) — Chemotaxis-convection-diffusion (CCD) have significant roles in medical, industrial, and geophysical areas, that is why research effort has been performed to understand the dynamics of the bacterial motility in suspension, studies through analytical, experimental, and numerical attempts previously were only for a flat free-surface of a suspension of chemotaxis bacteria in a shallow/deep chamber. We consider now a 3D CCD flow system with a deformed free-surface to explore the nature of instability by performing detailed stability analyses. Weakly nonlinear stability analysis has been carried out to determine the relative stability of the pattern formation at the onset of instability where Rayleigh number $Ra$, is the nonlinear control parameter. Nonlinear convection terms dominated the system beyond a critical $Ra$, which also depends on the critical wavenumber $k$ and Nusselt number $Nu$, as well as other parameters. We have investigated the issue of how the critical $Ra$, in this model varies with three different sets of parameters. Using the method of multiscales, a Ginzburg-Landau equation is derived from the Lorenz model (derived under the assumption of Bossinesq approximation), the solution of which helps to quantify the energy transport through $Nu$. 

1This work was supported by the National Natural Science Foundation of China under Grant Nos. 11702126 and 91752201.
9:31AM H13.00008 Linear thermal convection in an oscillatory fluid layer: A Floquet analysis of a sinusoidal, time-periodic basic flow. RUBEN AVILA, Facultad de Ingeniería, Universidad Nacional Autonoma de Mexico — The onset of thermal convection of a Boussinesq fluid confined in a plane layer with harmonic oscillation is investigated. The boundaries of the layer are parallel to the x-y plane of the Cartesian coordinate system. The temperature of the lower boundary is higher than the temperature of the upper boundary. The fluid layer oscillates around the y axis with a given amplitude and frequency. The basic velocity profile is sinusoidal and time-periodic. The flow with a linear basic temperature profile, ascends close to the hot boundary and descends close to the cold boundary. The basic harmonic is obtained numerically, and introduced as an analytical expression in the linearized equations of the vertical velocity, vorticity and temperature. The non-steady linear equations are formulated in terms of the parameters (the wave number, and the Rayleigh, Taylor and Prandtl numbers) that govern the system. The linear equations are solved (for a fluid with Prandtl number equal to 0.7) by a collocation method that is based on the Chebyshev polynomials. By using the Floquet theory, the thermal instability of the basic harmonic flow is studied. Curves of the critical values of the Rayleigh number and the wave number, as functions of the other parameters of the system are shown.

9:44AM H13.00009 Turbulent Mixing and Entrainment in a Buoyancy Driven Continuous Thermal Plume using Large-Eddy Simulations. SUDEEHER REDDY BHIMIREDDY, KIRAN BHAGANAGAR, University of Texas at San Antonio — Understanding the effects of extreme events such as wildland fires, volcanic eruptions are some of the critical issues the scientific community is facing now. For this purpose, a high-fidelity Large-Eddy Simulation has been developed to simulate thermal plumes with subcritical Froude’s number in a 4 km x 4 km x 7 km domain under idealized conditions. A fundamental fluid dynamics analysis has been performed to quantify the mean and turbulent characteristics of the thermal plumes. Additional 2-D simulations performed revealed that a well-defined head exists and travels with nearly constant velocity as long as it is attached to the plume stem. Due to intense turbulent mixing, 3-D thermal plume does not exhibit a well-defined head. To quantify the mixing and entrainment, 1st principle control volume approach based on tracking the plume interface has been used in both 2-D and 3-D cases. Using this, the interface we also study the plume ascent rate and half-widths based on both radial velocity and buoyancy profiles. Entrainment in 3-D thermal plumes is found to be significantly higher than in 2-D. The primary mechanism entraining the ambient fluid in 3-D is due to the instabilities at interface, whereas in 2-D, the centerline vorticity is found to be responsible for the mixing.

9:57AM H13.00010 Modeling of Round Buoyancy Driven Particle Clouds. ALI ALNAHIT, NIGEL KAYE, Associate Professor of Civil Engineering, ABDUL KHAN, Professor of Civil Engineering — A numerical model was developed to investigate the dynamics of round buoyancy driven particle clouds in a quiescent ambient. The developed model was validated and then applied to a range of test cases including releases of positively/negatively buoyant particles. The cloud was modeled using the standard Morton et al. (1956) entrainment assumption, and the flow field induced by the cloud was approximated as a Hill’s spherical vortex. The buoyancy of the cloud was calculated as the sum of the buoyancy contributed by all particles within the cloud. Individual particles were tracked using a particle tracking equation considering the forces acting on individual particles and the computed induced velocity field. Particle–particle interactions were modeled as both elastic and inelastic collisions. The turbulent dispersion of particles was also considered and estimated using a random walk model. The model was validated by comparing simulations with the experimental and numerical results of Wang et al. 2016. The limitations of the model were then discussed.

10:10AM H13.00011 Buoyancy distribution in a filling box segmented by a planar jet. NIGEL KAYE, Clemson University, NICHOLAS WILLIAMSON, University of Sydney — Results are presented from a theoretical and experimental investigation of the buoyancy distribution in a filling box in which the box is divided by a planar jet. The planar jet acts as a barrier that inhibits the transfer of buoyancy from the plume side to the non-plume side of the jet. Filling box theory is used to model the layer depth and total buoyancy on either side of the planar jet. Although the planar jet initially prevents the plume outflow from spreading into the non-plume side there is still some transfer of buoyancy due to entrainment of buoyant fluid into the planar jet that is then transferred to the non-plume side. The theoretical model is validated against a series of small scale salt bath experiments. The model well predicts the initial distribution of buoyancy between the plume and non-plume sides of the jet. However, the build-up of buoyancy on the plume side eventually bends the jet toward the non-plume side and the jet no longer provides a barrier. The time at which the model breaks down is well predicted by air-curtain theory. The results are discussed in the context of the potential for air curtains to inhibit smoke spread in compartment fires and potentially allowing increased evacuation times for occupants.

10:23AM H13.00012 Manipulation of Small-Scale Motions Induced by Self-Oscillating Reeds for Heat Transfer Enhancement. SOURABH JHA, ARI GLEZER, Georgia Institute of Technology — Low Reynolds (Re) number forced convection heat transport within the fin channels of air-cooled condensers are enhanced by deliberate formation of unsteady, small-scale vortical motions using aero-elastically fluttering thin-film reeds that span the channel height. These vortical motions substantially increase the local heat transfer coefficient at the channel walls and mixing between the wall thermal boundary layers and the cooler core flow. The flow mechanisms associated with production, advection and dissipation of these small-scale motions are investigated in a modular, high aspect ratio channel using micro-PIV, video imaging of the reed motion, and hot-wire anemometry. The global heat transfer enhancement is investigated in a modular heat sink prototype using temperature and pressure measurements. It is shown that the reed-induced small scale motions increase the turbulent kinetic energy of the flow even when the base flow undergoes transition to turbulence, leading to an increase in the local and global Nusselt number (Nu) that is sustained at higher Re and a minor relative increase in losses. While these losses depend primarily on the reed’s oscillation Strouhal number (St=fL/U) which is determined by the reed’s mass ratio (M*) and structural rigidity (I*) , because the global Nu depends only weakly on St, the losses associated with the presence of the reed can be strongly mitigated by reducing its St while maintaining the heat transfer enhancement.

Monday, November 25, 2019 8:00AM - 10:36AM — Session H14 Convection and Buoyancy-driven Flows: Turbulent Rayleigh-Benard
8:00AM H14.0001 Elusive transition to the ultimate regime of turbulent RBC: Dynamics of LSC in high-Ra cryogenic helium experiments1, MICHAL MACEK, PAVEL URBAN, PAVEL HANZELKA, TOM KRLK, VRA MUSILOV, The Czech Academy of Sciences, Charles University, Faculty of Mathematics and Physics — Non Oberbeck-Boussinesq (NOB) effects may increase the heat transfer efficiency of turbulent Rayleigh-Bénard convection (RBC), when the top plate temperature approaches the saturation vapor curve (SVC) even far away from the critical point of the working fluid. Our recent experimental study [1] using cryogenic He under conditions as close as possible to the Goettingen study using SF6 [2] argues that the claim of having observed the transition to Kraichnans ultimate \( Nu(Ra) \) scaling is likely due to NOB effects, and the important issue of transition to the ultimate state of RBC remains open. I will present here a detailed analysis of large-scale circulation (LSC) dynamics in the experiment [1]. I will discuss dependences of the Reynolds numbers associated with LSC circulation and sloshing and of the LSC reversal frequency on the position in the \( p-T \) diagram of \( ^4 \)He, in particular on the boundary layer asymmetry due to NOB conditions near the SVC. [1] P. Urban, P. Hanzelka, T. Krlk, M. Maccek, V. Musilov and L. Skrbek, Phys. Rev. E 99, 011101(R) (2019).

1Supported by NSF grant EAR-1620649.

8:13AM H14.00002 WITHDRAWN

8:26AM H14.00003 Velocity and Thermal Boundary Layer Equations for Turbulent Rayleigh-Bénard Convection1, EMILY SC CHING, H.S. LEUNG, Department of Physics, The Chinese University of Hong Kong, LUKAS ZWIRNER, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization — In turbulent Rayleigh-Bénard convection, the boundary layers (BLs) are non-steady with fluctuations, the time-averaged large-scale circulating velocity vanishes far away from the top and bottom plates, and the motion arises from buoyancy. There is no existing BL theory that successfully incorporates all these physical effects. We have derived a full set of BL equations for both the temperature and velocity fields from the Boussinesq equations for a two-dimensional fluid above a heated plate, accounting for all these physical effects. We use the commonly employed concepts of eddy viscosity and eddy thermal diffusivity to study fluctuations and propose a closure model to relate them to the stream function. This full set of BL equations enables us to obtain the time-averaged velocity and temperature BL profiles, in the form of similarity solutions, for general Prandtl number (\( Pr \)) in terms of two parameters \( k_1 \) and \( k_2 \) that measure the size of fluctuations. We have demonstrated that with a suitable choice of \( k_1 \) and \( k_2 \), our theoretical results are good approximations of both the time-averaged velocity and temperature profiles obtained in direct numerical simulations especially at low \( Pr \), for which theoretical results are challenging to obtain.

1OS and LZ acknowledge the financial support of the Deutsche Forschungsgemeinschaft (DFG) under grants Sh405/4-2 (Heisenberg fellowship) and Sh405/7 (Priority Programme SPP1881) respectively.

8:39AM H14.00004 Effects of Prandtl number in quasi-two-dimensional turbulent Rayleigh-Bnard convection1, XIAO-MING LI, JI-DONG HE, PENG HAO, SHI-DI HUANG, Department of Mechanics and Aerospace Engineering, South University of Science and Technology, Shenzhen, Guangdong 518055, China — We report an experimental study of the Prandtl (\( Pr \)) number effects on flow pattern and local temperature fluctuation in quasi-two-dimensional turbulent Rayleigh-Bénard convection. The experiments were conducted in four rectangular cells with same aspect ratio but different heights, the Rayleigh number \( Ra \) range (1e9 − 2e10) remains unchanged while \( Pr \) is varied from 11.6 to 157.4. The flow patterns visualized by the shadowgraph show that thermal plumes become more slender as \( Pr \) increases, and their organized-motion is more concentrated towards the sidewall. The mean flow strength, characterized by the Reynolds number \( Re \), becomes weaker with the increase of \( Pr \), i.e. \( Re(0.57Pr^{0.81}) \). It is further found that the temperature fluctuations in the center (\( \sigma_T/\Delta T \)) and near sidewall (\( \sigma_T/\Delta T \)) behave different, i.e. \( Pr<0.19Ra<0.28 \) and \( Pr>0.19Ra>0.20 \), respectively. This result quantitatively demonstrates that, as \( Pr \) increases, thermal plumes prefer to move along the sidewall rather than traveling through the center of the cell.

1This work was supported by the National Natural Science Foundation of China under Grant Nos. 11702128 and 91752201.

8:52AM H14.00005 Effect of slip boundary conditions on the heat flux and near-wall temperature equations in turbulent Rayleigh-Bénard convection1, XIAOZHOU HE, MAOJING HUANG, Harbin Institute of Technology(Shenzhen), YIN WANG, HONG KONG University of Science and Technology, YUN BAO, Sun Yat-Sen University — We present direct numerical simulations (DNS) of the heat transport and near-wall temperature profiles in turbulent Rayleigh-Bénard convection (RBC) with slip boundary conditions (BCs) on horizontal walls. The mean horizontal velocity on the wall is assumed as \( u_w = (b/L)(\partial u/\partial n)|_w \). Here \( L \) is the height of RBC sample, \( b \) is the slip length with \( b = 0 \) for no-slip BC and \( b \to \infty \) for free-slip BC. The simulations were for \( 0 \leq L/b \to \infty \) and the Prandtl numbers \( Pr = 4.3 \). It was found that the ratio of dimensionless heat flux, as expressed by the Nusselt number follows \( Nu/Nu_0 = 0.8 \times \tanh(100 \times b/L) + 1 \), where \( Nu_0 \) is the Nusselt number for \( b = 0 \). Considering the boundary layer fluctuations, we derived the equation \( \Theta(x) = \int_{-\infty}^{\infty}(1+2p^2q^2)^{-1}dp \) for the mean temperature profile \( \Theta(x) \) near the horizontal surface, where \( p = \Gamma(1+1/x)\Gamma(n-1/x)/\Gamma(n) \) with \( 2 \leq x \leq 3 \) depending on \( b/L \) and \( n > 1 \) for varying geometries of the convection sample.

1Supported by NSFC Grant No.11772111.

9:05AM H14.00006 A systematic investigation of the inverse cascade and flow speed scaling behavior in rapidly rotating Rayleigh-Benard convection1, MICHAEL CALKINS, University of Colorado, Boulder, STEFANO MAFFEI, University of Leeds, MITCHELL KROUSS, KEITH JULIEN, University of Colorado, Boulder — Rotating convection is a robust driving mechanism for the inverse kinetic energy cascade, which is characterized by the transfer of energy from small-scale convective motions to large-scale motions. One of the consequences of this process is the formation of so-called large-scale vortices (LSVs) that fill the entire flow domain. However, the domain of existence for these vortices and the behavior of their saturated amplitude is not well-constrained. A systematic set of simulations of the asymptotically-reduced governing equations show that: (1) the presence of LSVs is characterized by a critical small-scale Reynolds number over a wide range of Prandtl numbers; (2) the amplitude of the LSVs scales predictively with the horizontal dimensions of the flow domain; and (3) the amplitude of the LSVs saturates with increasing Rayleigh number. Furthermore, we find that LSVs are present in flow regimes not previously known to harbor them. A scaling law is developed for the convective Reynolds number as a function of Rayleigh number and Prandtl number, which can be used to predict the presence of LSVs.

1This work was funded by NSF grant EAR-1620649.
In summary, four local Nu-scalings are obtained: 1. buffer layer thickness and the ejecting velocity as large scale circulation velocity, the balance of inertia and buoyancy yields $\text{Nu}$ the similarity of the momentum and thermal boundary layer. In plume-ejecting region, after choosing the proper scale of plume scale as the intermittent boundary layer dynamics. Various strategies for improving the efficiency of statistical convergence will also be presented.

We acknowledge support by the Priority Programme SPP 1881 of the Deutsche Forschungsgemeinschaft and an award of computer time provided by the INCITE program. This research used resources of the ALCF at ANL, which is supported by the DOE under contract DE-AC02-06CH11357.

We present the dependence of the heat transport (the Nusselt number) and momentum transport (the Reynolds number) as expected, the energy transfer differs significantly between the bulk and close to the wall. At large Rayleigh numbers, the energy input into the superstructures is primarily balanced by a direct energy transfer to smaller scales in the bulk. Here, the energy transfer acts as an effective dissipation for the superstructures. However, close to the wall the energy transfer is more complex and may even drive the superstructures. Besides a characterization, these results can help to develop an effective description of turbulent superstructures in convection.

Inhomogeneous Nu distribution along horizontal plate, with four flow regimes, i.e. corner-roll, jet-impingement, wind-shearing and plume-ejecting. In corner-roll, Nu scaling follows a modified mixing zone model, with a scale correction $r \sim Ra^{-0.65}$. In jet-impingement region, a streamwise momentum similarity yields an Nu distribution: $Nu_{ji} = 0.2Ra^{1/4} \exp(-1.2(x-x_{eo})/L_{ji})$. In wind-shearing region, side wall does not affect the similarity of the momentum and thermal boundary layer. In plume-ejecting region, after choosing the proper scale of plume scale as the buffer layer thickness and the ejecting velocity as large scale circulation velocity, the balance of inertia and buoyancy yields $Nu_{pe} \sim Ra^{1/369}$. In summary, four local Nu-scalings are obtained: 1/3, 0.30, 0.262, and 0.369, and the global Nu scaling is then obtained by adding all four with multiplying them with their spatial domain sizes. This model provides an alternative interpretation for Nu-scaling transition.

This work was supported in part by the Research Grants Council of Hong Kong SAR and the National Natural Science Foundation of China.

1This work was supported by National Nature Science Fund 11221062, 11452002 and by MOST 973 project 2009CB724100.

1Supported by National Nature Science Fund 11221062, 11452002 and by MOST 973 project 2009CB724100.

1This work was supported in part by the Research Grants Council of Hong Kong SAR and the National Natural Science Foundation of China.

1This work was supported by the National Natural Science Foundation of China under Grant Nos. 11702128 and 91752201.
10:23AM H14.00012 Near free-fall oscillatory velocities in liquid metal rotating convection. TOBIAS VOCT, HZDR, SUSANNE HORN, Coventry University, JONATHAN AURNOU, UCLA, HELMHOLTZ-ZENTRUM DRESDEN-ROSENTORF TEAM, UNIVERSITY OF CALIFORNIA, LOS ANGELES TEAM, COVENTRY UNIVERSITY TEAM — The geomagnetic field is induced by liquid metal flow inside Earths outer core as a self-excited dynamo. Buoyancy drives the liquid metal flow because the iron rich core is cooling from its primordial state through heat loss to the mantle. The rotation of the Earth and Lorentz forces alter the resulting convective flow. However, the detailed flow topology is largely unknown. Here we will investigate the effect of rotation on a low Prandtl number thermal convection by means of laboratory experiments and DNS. Therefore, we consider a rotating Rayleigh-Bénard convection setup in an upright cylindrical vessel of aspect ratio $\Gamma = D/H \approx 2$. We investigate supercriticalities in the range of $1 < \text{Ra} < 20$ and Ekman numbers $4 \times 10^{-5} \leq \nu \leq 2 \times 10^{-6}$ in liquid gallium at $Pr = 0.03$. By means of ultrasound-Doppler velocity measurements, we find that oscillatory convection generates velocities approaching the freefall velocity. Multi-modal bulk oscillations dominate the vertical velocity field over the whole range of supercriticalities investigated. Additionally, coherent mean zonal flows and time-mean helicity suggesting that these oscillatory flows can be relevant for dynamo action.

Monday, November 25, 2019 8:00AM - 10:36AM — Session H15 Turbulent Boundary Layers: Roughness and Atmospheric Turbulence

8:00AM H15.00001 Altered large-scale structures in turbulent boundary layers formed over drag-increasing riblets. RYAN NEWTON, DANIEL CHUNG, NICHOLAS HUTCHINS, University of Melbourne — Riblet drag reduction only occurs within a small, viscous-scaled riblet spacing envelope, outside of which drag is significantly increased. In this study, non-optimal, drag-increasing trapezoidal riblet surfaces, with a 30° tip angle and height of 0.5s are experimentally investigated in the regime where the viscous-scaled riblet spacing is large ($40 < s^+ < 250$). The motivation behind this is to understand the cause of the drag increase in this regime. Results indicate that riblets with a large $s^+$ provide a considerable drag penalty, which appears to asymptote to fully rough behavior as $s^+$ increases. An analysis of the turbulence close to the surface reveals that there is little evidence of either lodgment of near-wall turbulence in the riblet grooves, or of a Kelvin–Helmholtz instability above the riblet crests. This suggests that the two previously hypothesized mechanisms for the breakdown in drag reduction at $s^+ \approx 20$, do not play a role at large $s^+$. In addition, energy spectra show that the riblets greatly diminish the footprint of large-scale turbulent motions in the near-wall and logarithmic regions, questioning the presence of a typical log layer and the validity of outer-layer similarity for these cases.

8:13AM H15.00002 Scale interactions in velocity and pressure over urban-like roughness. MANUEL FERREIRA, BHARATHRAM GANAPTHISUBRAMANI, University of Southampton — An integral description of a boundary layer developing over a large-cube array is obtained using velocity data to reconstruct the underlying pressure field via 2D-TH. Coupled-statistics of the pressure forces acting on a target roughness element provide insight into the relevant mechanisms responsible for surface drag. This is complemented by conditional analysis and extended POD of the pressure field based on velocity modes to further understand the velocity-pressure interrelations. Coherent motions, at different scales, leave a strong imprint on the pressure field. Larger turbulent features dominate pressure variance, but their direct contribution to surface drag appears to be mitigated by the relative size of the roughness obstacles that are considerably smaller. Pressure waves induced by the passage of alternating high and low-momentum regions evenly affect the flow field over a wide region, coupling the forces on the windward and leeward sides of the cube which, in turn, partially cancel each other out. This suggests that uncorrelated, intermediate and small-scale pressure fluctuations are more important to the drag force variance. Large-scale structures are nonetheless significant for the role they play in modulating the small-scale pressure events in the near wall region.

8:26AM H15.00003 Post fully rough regime in turbulent pipe flow. DANIEL CRUZ, HAMIDREZA ANBARLOOEI, FABIO RAMOS, CECILIA MAGESKI, ATILA FREIRE, Federal University of Rio de Janeiro — The phenomenological model of Gioia and Chakraborty (2006) tries to relate the friction and energy spectrum in the friction layers. In this model, the momentum transfer by eddies at a specified surface above the wall shear stress. At low Reynolds numbers, the dominant eddies are in the size of the Kolmogorov length scale (Blasius regime), while at high Reynolds numbers the dominant eddies are in the size of the roughness elements (fully rough regime). The present authors recently showed that instead of a simple transition between these two extremes, there is a third regime where the effects of the wall (through attached eddies) dominate. This results in a new power-law friction regime between the Blasius and fully rough regimes. The effects of the roughness geometry on turbulence are studied experimentally in the present work. Beside the common sand-roughed surfaces, some well geometrically defined rough walls have also been examined. As expected, by increasing the Reynolds number, the fully rough regime appears for the latter type of roughness. However, a further increase in the Reynolds reveals an unexpected behavior, where friction starts to decrease again in a power-law manner (with the same trend as observed for the third regime).

8:39AM H15.00004 The influence of regular and random roughness on outer-layer similarity. MICHAEL SCHULTZ, United States Naval Academy, KRISTOFER WOMACK, Johns Hopkins University, RALPH VOLINO, United States Naval Academy, CHARLES MENEVEAU, Johns Hopkins University — An experimental investigation was carried out on rough-wall, turbulent boundary layers with regular and random roughness element arrangements. Varying planform densities of truncated cone roughness elements in square staggered patterns and random arrangements were investigated. Velocity statistics were measured via two-component laser Doppler velocimetry and stereo particle image velocimetry. Differences in the mean streamwise velocity profiles recorded over the regular arrangements were confined to within one roughness height above the roughness crests whereas the random cases exhibited significant variation across the span that persisted into the outer layer. Evidence is shown that the differences result from low momentum pathways (LMPs) and high momentum pathways (HMPs) over the random surfaces which are not observed over the regular surfaces. The LMPs and HMPs were marked by elevated and depressed Reynolds shear stress as seen by previous investigators. However, previous studies had systematic or regular surface topography which they linked to the secondary flows. The present results indicate that secondary flows may develop even in the absence of these surface characteristics.

The authors gratefully acknowledge financial support from the National Defense Science and Engineering Graduate Fellowship (K.W.), the US Office of Naval Research (M.S. and R.J.), and NSF grant CBET-1738918 (C.M.).
The form-induced velocity affects the budget of $\overline{\theta'\theta'}$, more than that of $\overline{u'u'}$.

1This research was supported by NSERC.
10:10AM H15.00011 Moving horizon estimation of turbulent velocity fields in the atmospheric boundary layer using lidar measurements and large eddy simulations\(^1\), PIETER BAUWEREAUTS, JOHAN MEYERS, KU Leuven — Pulsed lidar sensors measure the line-of-sight (LOS) projected wind velocity in the atmospheric boundary layer at a range of locations along the LOS spanning several kilometres. Despite the vast amount of measurement information, the data remains sparse and unidirectional. To reconstruct a full 3D velocity field, we employ a non-linear moving horizon estimation (or 4D-Var) approach, using large-eddy simulations (LES) as a state space model. The cost function, which represents the relative probability of the state space trajectories, is comprised of two terms, the first term regularizes the optimization based on prior statistical knowledge of the velocity field fluctuations embedded in the 2-point covariance tensor of the atmospheric boundary layer. To this end, the covariance tensor is averaged offline over a sufficiently long time window, to ensure statistical convergence. The second part penalizes the difference between the LES and real observations over the time horizon. Instead of using experimental data, we use a fine-grid LES simulation as a virtual reality, creating a controlled and flexible testing environment. The methodology is demonstrated on two measurement setups, a standard plan-position indicator scanning mode and a 3D trajectory based on a Lissajous curve.

1Supported by the Agency for Innovation and Entrepreneurship through research grant no. 141689.

10:23AM H15.00012 Resolving near-wall particle dynamics for turbulence-induced saltation and dispersion , TAEHOON KIM, SAMUEL HAMERMESH, RUI NI, Johns Hopkins University — The Stanley-Corssin wind tunnel at Johns Hopkins University has recently been repurposed for studying sediment transport over an erodible surface. To resolve the near-wall particle dynamics, a hybrid technique that uses both the back illumination and scattering light illumination was adopted to measure the particle dynamics over a wide range of scales. In particular, it allows us to resolve particle motion across the near-wall saltation layer using our in-house particle tracking code. From this experiment, the entire time history of particle behaviors can be acquired. In this presentation, we will discuss the statistics of near-wall particle motion and its relationship with the near-wall turbulence.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H16 Focus Session: Exascale Computations of Complex Turbulent Flows I

8:00AM H16.00001 Thoughts on Very Large Turbulence Simulations , PHILIPPE SPALART, Boeing Commercial Airlines — The Reynolds number in channel DNS rose by a factor of 29 since the KMM paper of 1987. The imminent step from about 10\(^{4}\) FLOPS to 10\(^{7}\) FLOPS allows a factor of about 3. If we elect to simulate the same flow. Turbulence created three challenges: its results conflict with our semi-theory of wall-bounded flows. First, even with this factor of 29, a true logarithmic layer is not found. Second, the Reynolds stresses do not have a plateau in the (near) log layer. Third, they are not independent of the flow Reynolds number at a fixed \(y^+\). In fact, all these properties are implied by classical theory and satisfied by turbulence models of conventional type. This has greatly limited the contribution of DNS to RANS models. The complaint in engineering is over smooth-body separation and reattachment. DNS of such flows at two Reynolds numbers are most desirable, but defining the same flow at different Reynolds numbers is non-trivial because of the incoming boundary layer thickness. A strategic community decision is whether simulations can be free-standing, or experimental confirmation is needed. The use of simulation data, be it physical understanding, engineering tools built with classical thinking, or tools built with Machine Learning, is challenging. For this third option, a sound definition of the mission is sorely needed. Also in high demand are non-DNS turbulence-resolving simulations, including Wall-Modeled LES and DES, as are simulations over complex geometries, say a Formula-1 car. Such simulations have had poor grids, and automatic adaptation is needed. Grid generation, even non-adaptive, could create a bottle-neck on massively-parallel machines. Grid convergence is not achieved over full geometries, even for RANS. Research can be improving SGS and Wall Models using DNS flow fields or exercising the non-DNS tools at high Reynolds numbers. It is difficult to define challenges that are clear, are attractive in an academic sense, and will illustrate the value of having 1 exaFLOPS.

8:13AM H16.00002 Tractability of LES for complex flows\(^1\), SANJEEB BOSE, KAN WANG, CHRIS IVEY, FRANK HAM, Cascade Technologies — To date, the use of large-eddy simulations (LES) has often been restricted due to relatively large computational expense (leading to intraceltably long wall clock times) and challenges in the generation of suitable grids for complex geometries. Advances in scalable mesh generation, nonlinearly stable numerical methods and wall modeling have substantially reduced the computational cost of performing LES of flows with either complex physics and/or geometries. We will discuss the computational performance and scalability of the flow solver charLES on petascale architectures and the prospects of using LES within engineering design environments where higher simulation throughput is required. The cost and fidelity will be assessed in the context of some benchmark LES, including the prediction of transonic compressor stall (NASA Rotor 37). By leveraging massively parallel petascale computing platforms, it will be shown that such calculations can be completed within hours of wall clock time.

1This work has been supported by computational time provided through the DOE INCITE program (at ALCF) and at OLCF.

8:26AM H16.00003 Exascale Turbulence Simulations: From Fundamental Flows to Flight Scale Aerodynamics\(^1\), KENNETH JANSEN, RICCARDO BALIN, JOHN EVANS, University of Colorado Boulder, PHILIPPE SPALART, The Boeing Corporation — This talk will provide an update on two Argonne Early Science Program (ESP) projects. The first is a Simulation ESP where prior delayed detached eddy simulations of flow control on a vertical tail at a chord Reynolds number of 325k were validated against experiments being extended to flight scale (53 times higher Reynolds number). The second is Data and Learning ESP focused on: 1) data compression, 2) turbulence modeling improvement from machine learning, 3) uncertainty quantification and multi-fidelity modeling, and 4) in situ data analytics. Progress made towards these ambitious goals will be shared as well as future plans and needed developments.

1This work is supported not only by Argonne National Labs through the aforementioned ESP projects but also by the National Science Foundation (NSF), award number CBET-1710670 and by the National Aeronautics and Space Administration (NASA), award number 80NSSC18M0147. Resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility, and of the NASA HECC facilities were used.
This research used resources of the Argonne Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357.

1Funded by the ERC COTURB project. Part of the computational time was provided by the PRACE TIER-0 project “TREC”.

8:52AM H16.00005 Toward Simulating Turbulent Wall-Bounded Flows at High Reynolds Numbers on Exascale Platforms1, RAMESH BALAKRISHNAN, Argonne National Laboratory, PAUL FISCHER, University of Illinois at Urbana-Champaign — Given that wall resolved LES (WRLES) of high Reynolds number flows will continue to be intractable on even exascale computing platforms, there is considerable effort to make hybrid RANS/LES (HRLES) methods the tool of choice for predictive simulations of high-Re separated flows. The fidelity of HRLES depends on the ability of the subgrid model to account for the effects of the filtered scales on the resolved scales, and on the numerical schemes that are used to evolve the Navier-Stokes equations. While higher-order numerical schemes, with low dissipation/dispersion errors (and higher arithmetic intensity), are commonly used for canonical flow simulations (on simpler geometries), the bulk of the high-Re flow simulations on complex geometry still employ nominally second-order accurate schemes in structured/unstructured flow solvers. Hence, there is a need for better sub-grid models that can improve the predictive capability of both the higher-order flow solvers and existing second-order accurate flow solvers, for simulating turbulent separated flows. This talk is about ongoing efforts to develop HRLES subgrid models from DNS/WRLES simulations of canonical flows over curved and sharply discontinuous surfaces and their ability to predict high-Re flows on current petascale platforms.

1This research used resources of the Argonne Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357.

8:26AM H16.00006 Toward Simulating Turbulent Wall-Bounded Flows at High Reynolds Numbers on Exascale Platforms1, RAMESH BALAKRISHNAN, Argonne National Laboratory, PAUL FISCHER, University of Illinois at Urbana-Champaign — Given that wall resolved LES (WRLES) of high Reynolds number flows will continue to be intractable on even exascale computing platforms, there is considerable effort to make hybrid RANS/LES (HRLES) methods the tool of choice for predictive simulations of high-Re separated flows. The fidelity of HRLES depends on the ability of the subgrid model to account for the effects of the filtered scales on the resolved scales, and on the numerical schemes that are used to evolve the Navier-Stokes equations. While higher-order numerical schemes, with low dissipation/dispersion errors (and higher arithmetic intensity), are commonly used for canonical flow simulations (on simpler geometries), the bulk of the high-Re flow simulations on complex geometry still employ nominally second-order accurate schemes in structured/unstructured flow solvers. Hence, there is a need for better sub-grid models that can improve the predictive capability of both the higher-order flow solvers and existing second-order accurate flow solvers, for simulating turbulent separated flows. This talk is about ongoing efforts to develop HRLES subgrid models from DNS/WRLES simulations of canonical flows over curved and sharply discontinuous surfaces and their ability to predict high-Re flows on current petascale platforms.

1This research used resources of the Argonne Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357.

9:05AM H16.00006 Wall-resolved LES of a complex turbulent flow, MUJEEB MALIK, NASA Langley Research Center, ALI UZUN, National Institute of Aerospace — Implicit, wall-resolved large eddy simulation, using a fourth-order compact difference scheme, is performed for a relatively high Reynolds number flow involving shock-induced flow separation. This is perhaps one of the most ambitious such simulations employing 24 billion grid points, pushing the boundary of flow simulations on high performance computing hardware. The particular case selected for this simulation is that of the well-known Bachalo-Johnson experiment conducted on a cylindrical body with an axisymmetric bump, which involves transonic shock-induced boundary layer separation with subsequent reattachment downstream of the bump. The Reynolds number based on the hump chord is 2.763 million. The relatively high Reynolds number of the test case makes the wall-resolved simulation very challenging, requiring billions of grid points. We will discuss the various issues and challenges encountered during the course of this research. Comparison of the simulation results is made with the available experimental measurements, with a view toward assessing the predictive capability of the simulations. Work supported by NASA’s Transformational Tools and Technologies Project.

9:18AM H16.00007 Wall-Modeled Large Eddy Simulation (WMLES) of High-Lift Aircraft1, KONRAD GOC, SANJEEB BOSE, PARVIZ MOIN, Stanford University — Wall-Modeled Large Eddy Simulations with an equilibrium wall model (WMLES) are carried out on the JAXA high-lift model at various angles of attack. The configuration features the geometric complexity of deployed slats, flaps, and associated bracketry with flow in the low Mach, high Reynolds number regime (e.g., takeoff conditions). Salient flow features arising from smooth-body and geometrically-imposed separation are captured. The lift curve slope predicted by WMLES is well-characterized near the stall flight condition. Good agreement with experimental sectional pressure measurements is obtained. Calculations that include the wind tunnel walls and half-model mount are compared to uncorrected experimental data to mitigate wind tunnel correction uncertainty. Simulations were performed using the CharLES code at core-hour costs comparable to some RANS calculations of the same configuration (Rumsey, 2018).

1Computer resources were provided through the INCITE Program of the Department of Energy and research funding through NASA and Boeing Research & Technology (BR&T).

9:31AM H16.00008 Harnessing Exascale Platforms to Predict Shock-Wave / Boundary-Layer Interaction1, JONATHAN POGGIE, Purdue University — Separated shock-wave / boundary-layer interactions occur in a broad flight regime, from transonic commercial aircraft to hypersonic space-access vehicles. These interactions tend to be intensely unsteady, with frequency content from fine-grained turbulence to low-frequency (~ 10 Hz) separation bubble breathing. The presence of this unsteadiness can cause problems with flight control and structural fatigue on aircraft. Simulation of this important phenomenon is extremely challenging because of the need to capture such a broad range of scales. Identification of the physical mechanisms underlying the unsteadiness may help guide the development of simplified models that capture the essential features of the interactions. To this end, our research group has been investigating separated compression ramp flows, and studied their selective response to disturbances in the incoming turbulent boundary layer. In implicit large-eddy simulations, conditional averaging identified a perturbation velocity profile associated with separation motion, and forcing in the upstream boundary layer with this particular form was found to drive large-scale separation unsteadiness downstream.

1This work has been supported by a DoE INCITE Award and AFOSR Grant FA9550-17-1-0153.
9:44AM H16.00009 Shock-induced transition and heating in hypersonic boundary layers$^1$, LIN FU, MICHAEL KARP, Center for Turbulence Research, Stanford University, Stanford CA 94305. SANJEEB T. BOSE, Cascade Technologies Inc., Palo Alto, CA 94303. PARVIZ MOIN, JAVIER URZAY, Center for Turbulence Research, Stanford University, Stanford CA 94305 — The interaction of an incident shock wave with a Mach-6 undisturbed laminar boundary layer is addressed using DNS and equilibrium wall-modeled LES (WMLES). The wall temperature is cold compared to the free-stream stagnation temperature, such that the mean temperature profile develops a peak near the wall due to viscous heating. The consequences of the interaction are that the boundary layer transitions to turbulence downstream of the shock impingement point, and that transition causes a localized significant increase in the Stanton number and skin-friction coefficient. The peak thermomechanical load increases approximately linearly with the incidence angle. WMLES prompts transition and peak heating, delays separation, and shortens the separation bubble. WMLES provides predictions of DNS peak loads within 10% at 150 times lower computational cost. In the fully-turbulent boundary layer, WMLES agrees well with DNS for the Reynolds-analogy factor (Chi and Spalding, 1966), the mean velocity and temperature profiles, including the temperature peak, and the temperature/velocity correlations.

$^1$U.S. Air Force Office of Scientific Research (AFOSR) No. 1194592-1-TAAHO, INCITE

9:57AM H16.00010 Large-Eddy Simulations of turbomachinery flows$^1$, KOEN HILLEWAERT, Université de Liège, MICHEL RASQUIN, THOMAS TOULORGE, Cenaeo — Argo is developed for scale-resolving simulations in turbomachinery in view of generating reference data for RANS models as well as the prediction of off-design performance. Based on the Discontinuous Galerkin Method, low dispersion and dissipation commensurate with DNS and LES is maintained on an unstructured mesh. Recently, work is ongoing on wall modeling in view of full machine computations. The development of turbulence models in turbomachinery is hindered by the lack of detailed reference data, as usually only experimental data are available. Due to the continuous increase in computational power, it has become possible to generate detailed numerical databases in fully controlled conditions for industrially relevant conditions. The generation and exploitation of such data sets is the aim of the European project HiFiTurb. This contribution focuses on a variant of the axisymmetric transonic bump of Bacchalo and Johnson, storing all of the terms relevant to RANS, as well as time-resolved data in the boundary layer in view of improving modeling for shock-boundary layer interaction conditions.

$^1$We gratefully acknowledge the allocation by the Argonne Leadership Computing Facility in the INCITE 2019 program, as well those granted on the Tier-1 supercomputer of the Fédération Wallonie-Bruxelles, infrastructure funded by the Walloon Region under the grant agreement n1117545. Cenaeo also benefits from financing through the European union research project HiFiTurb.

10:10AM H16.00011 Large-Eddy Simulations of turbomachinery flows: from wall-resolved academic configurations to wall-modeled industrial geometries, , FLORENT DUCHAINE, LUIS SEGUI, JEROME DE LABORDERIE, NICOLAS ODIER, JEROME DOMBARD, LAURENT GICQUEL, Cercacs, CFD TEAM — LES has been shown to be a promising tool to tackle turbomachinery challenges induced by high Reynolds and Mach numbers and complex flow physics. The CPU cost is however identified as the reason why existing LES of turbomachinery flows concern simplified configurations. CERFACS has extended the capability of the reactive LES solver AVBP for turbomachinery applications. The developments and validations have concerned the application of accurate boundary conditions at inlets and outlets as well as the numerical treatment of the rotor/stator interface compliant with LES requirements. After a brief description of the flow solver, the presentation will focus on the results and insights obtained on two configurations. The first one is the high-pressure turbine cascade LS89 for which the MUR239 operating point is still today a challenge to simulate accurately. To address this academic aerothermal case, a wall-resolved approach is used with a high-order numerical scheme. The second configuration of interest is the 3.5 stages high-pressure axial compressor CREATE. This simulation, corresponding to one of the first wall-modeled LES of such a complex machine, has shown very promising results by comparison with experimental data.

10:23AM H16.00012 Challenges and progress in adaptation of RANS models for Scale Resolving Simulations (SRS) of Turbulence , PEDRAM TAZRAEI, Mechanical Engineering, Texas AM University, SHARATH GIRIMAJI, Ocean Engineering and Aerospace Engineering, Texas AM University — Even with the advent of exascale computing, many complex turbulent flows require some degree of modeling to enable predictive calculations of real-life applications. Scale resolving simulations (SRS) capable of yielding the best possible results at various and varying degrees of resolution are ideally suited for these computations. In general, SRS models can be broadly classified into zonal and bridging methods depending on the manner in which scale resolution is achieved. While the need for SRS approach is compelling, there is no clear consensus on closure model development thus far. In this presentation, we present a formal framework for adapting well-tested RANS (Reynolds-Averaged Navier-Stokes) models for SRS sub-grid stress computations. Various challenges such as commutation error, appropriate fixed-point behavior, near-wall closures are identified. Reasonably rigorous theoretical techniques to address the above issues in SRS modeling context are proposed. Asymptotic approach of SRS toward DNS (direct numerical simulations) in the limit of cut-off length scale approaching Kolmogorov length scale is also examined.

Monday, November 25, 2019 8:00AM - 10:36AM —
Session H17 Minisymposium: Machine Learning in Fluid Mechanics 4c4 - Jeff Eldredge, University of California, Los Angeles

8:00AM H17.00001 Opportunities for Machine Learning in Fluid Mechanics$^1$, MICHAEL BRENNER, Harvard University and Google Research — There are tremendous opportunities to use recent advances in machine learning and artificial intelligence to advance fluid mechanics as a discipline. This talk will give an overview of these opportunities, including (i) making scientific discoveries, such as the discovery novel flow phenomena; (ii) defining new representation of dynamical flows that make numerical solvers more efficient; (iii) the design of novel methods for experimental imaging and characterization; and (iv) development of novel coarse-grained solvers for the Navier Stokes equations. I will summarize the advances in Machine Learning that have made these opportunities possible and include some recent examples from our own work.

$^1$National Science Foundation and Simons Foundation
8:26AM H17.00002 Interpretable and Generalizable Machine Learning for Fluid Mechanics1, STEVEN BRUNTON, University of Washington — Many tasks in fluid mechanics, such as design optimization and control, are challenging because fluids are nonlinear and exhibit a large range of scales in both space and time. This range of scales necessitates exceedingly high-dimensional measurements and computational discretization to resolve all relevant features, resulting in vast data sets and time-intensive computations. Indeed, fluid dynamics is one of the original big data fields, and many high-performance computing architectures, experimental measurement techniques, and advanced data processing and visualization algorithms were driven by decades of research in fluid mechanics. Machine learning constitutes a growing set of powerful techniques to extract patterns and build models from this data, complementing the existing theoretical, numerical, and experimental efforts in fluid mechanics. In this talk, we will explore current goals and opportunities for machine learning in fluid mechanics, and we will highlight a number of recent technical advances. Because fluid dynamics is central to transportation, health, and defense systems, we will emphasize the importance of machine learning solutions that are interpretable, explainable, generalizable, and that respect known physics.

1Army Research Office PECASE (W911NF-19-1-0045; Program Manager: Dr. Matthew Munson), and Air Force Office of Scientific Research YIP (FA9550-18-1-0200; Program Manager: Dr. Gregg Abate)

8:52AM H17.00003 Machine learning for Predictive Turbulence Modeling : A Cautiously Optimistic Perspective1, KARTHIK DURAIYAM, University of Michigan — Machine learning has shown promise in describing, reconstructing or even predicting properties of a given system, given large amounts of relevant data. This talk focuses on how one can construct data-augmented models for turbulent flows that can learn from different systems, and transfer this modeling knowledge to make predictions in other systems in a non-parametric context. This defines a paradigm of transfer learning in the sense that the learning should target global rules - rather than problem-specific information - that is common to a class of systems that share similar physics. In the limit of finite (big or small) data, this requires the enforcement of a variety of physical, physics-inspired and empirically-known constraints. We embed learning within a context within PDE models in an integrating fashion, thus enforcing consistency between the learning and model construction. Examples of the enforcement of hard and soft constraints will be provided. These hybrid models are trained across different systems that are representative of the underlying model discrepancy, yielding predictions on unseen problems with quantified error bounds. Algorithmic, physical and data-related challenges will be discussed toward the end of achieving truly robust and generalizable models.

1Supported by the National Science Foundation & the Office of Naval Research

9:18AM H17.00004 Deep Reinforcement Learning for Flow Control1, PETROS KOUMOUT-SAKOS, ETH Zurich — Reinforcement Learning (RL) is a mathematical framework for problem solving that implies goal-directed interactions of an agent with its environment. The agent has a repertoire of actions, perceives states and learns a policy from its experiences in the large of rewards. Deep recurrent neural networks can be used to encode the state-action policy of the agent and effective training implies a selective choice of experiences. RL does not require a model of the dynamics, such as a Markov transition model, and relies instead on repeated interactions of the agent with the environment. I consider that this approximation makes it highly suitable for complex problems in fluid dynamics, albeit, presently, only when significant computing power or experimental automation is possible. Deep RL has been very successful in games (from backgammon to video games and the game of Go) and robotics, but remains rather unexplored in fluid mechanics. I hope to illustrate some of the strengths and limitations of Deep RL for flow control using simulations of perching and gliding bodies as well collective fish swimming.

1European Research Council Advanced Investigator Award 341117, Swiss National Science Foundation, Swiss National Supercomputing Centre (CSCS)

9:44AM H17.00005 Classifying Flows using Neural Networks EVA KANSO1, University of Southern California — Swimming organisms and robotic vehicles create long-lived flow disturbances that can in principle be detected and even exploited for tracking and navigation. Experimental evidence suggests that many aquatic organisms, from mate seekers to hungry predators, respond to specific hydrodynamic cues created by their respective preys. However, the exact features that make these flows distinguishable and the sensory measurements and layouts that are needed to detect them remain elusive. Here, we consider the inverse problem of classifying flow patterns from local sensory measurements. Specifically, we train neural networks to classify flow patterns by relying on flow sensors that measure a time history of the local flow signal at the sensor location. We systematically investigate the network performance for distinct types of sensory measurements: vorticity, flow velocities parallel and transverse to the direction of flow propagation, and flow speed. We show that the networks trained using transverse velocity outperform other networks, even when subjected to aggressive data corruption. We then train the network to classify flow patterns from instantaneous (one time) measurements, using a spatially-distributed array of sensors. The networks based on the spatially-distributed sensory arrays exhibit remarkable accuracy in flow classification, even when only a handful of sensors are active. We conclude by commenting on the advantages and limitations of these models for flow detection and classification, and we discuss how these results lay the groundwork for developing combined data-driven and physics-based models for flow sensing using distributed sensory arrays.


10:00AM H17.00006 Differentiable Fluid Simulations for Deep Learning NILS THURECK, Technical University of Munich — In this talk I will focus on the possibilities that arise from recent advances in the area of deep learning for accelerating and improving physics simulations. In this context, the Navier-Stokes equations represent an interesting and challenging advection-diffusion PDE that poses a variety of challenges for deep learning methods. In particular, I will focus on highlighting how differentiable fluid solvers can guide deep learning processes, and support finding desirable solutions. The existing numerical methods for efficient solvers can be leveraged within learning tasks to provide crucial information in the form of reliable gradients to update the weights of a neural networks. Interestingly, it turns out to be beneficial to combine supervised and unsupervised approaches. The former poses a much simpler learning task by providing explicit reference data that is typically pre-computed. Unsupervised learning on the other hand can provide gradients for a larger space of states that are only encountered during training runs. Here, differentiable solvers are particularly powerful to, e.g., provide neural networks with feedback about how inferred solutions influence the long-term behavior of a physical model. I will demonstrate this concept with several example solutions of coupled interactions of fluids and physical systems. Learnt solvers not only represents a valid physical system, but also an end-to-end training process for the field of physics-based deep learning. I will conclude by discussing current limitations and by giving an outlook about promising future directions.
8:00AM H18.00001 Optimum heart rate for brain-heart hemodynamic coupling and its clinical relevance for neurodegenerative diseases1, ARIAN AGHILINEJAD, FAISAL AMLANI, Department of Aerospace and Mechanical Engineering, University of Southern California, Los Angeles, CA, USA, KEVIN KING, Huntington Medical Research Institutes, Advanced Imaging Center, Pasadena, CA, USA, NIEMA PAHLEVAN, Department of Aerospace and Mechanical Engineering, University of Southern California, Los Angeles, CA, USA — Neurodegenerative diseases such as Alzheimer’s dementia have reached an epidemic proportion with a significant impact on public health. Disproportionate age-related stiffening of the aorta compared with the carotid arteries is theorized to reduce the protective impedance mismatch at the aorta-carotid interface and affect the regulation of cerebral hemodynamics. Recent clinical data has indicated the existence of a potential link between arterial stiffening, as measured by arch pulse wave velocity (PWV), and Alzheimer’s disease. In this study, we used an energy-based analysis of hemodynamic waves to quantify the effect of aortic arch stiffening on pulsatile hemodynamic transmission to cerebral vasculature. A one-dimensional blood flow model of the human vascular network was used to simulate pressure and flow wave propagations in the systemic circulation. Our results show there exists an optimum wave condition—occurring near normal human heart rates—that creates an optimum hemodynamic state for heart-brain coupling and minimizes pulsatile energy transmission to the brain. This is clinically important since excessive pulsatile energy transmission to microvasculature causes end-organ damage that may promote progression of neurodegenerative disease such as Alzheimer’s dementia.

1Arian Aghilinejad gratefully acknowledges Viterbi School of Engineering fellowship support from the University of Southern California (USC).

8:13AM H18.00002 Modelling Direct Brain Cooling for Acute Ischaemic Stroke with the Vascular-Porous Model 1, LUKE FULFORD, Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, IAN MARSHALL, PETER ANDREWS, Centre for Clinical Brain Sciences, University of Edinburgh, PRASHANT VALLURI, Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh — Ischaemic stroke is a major cause of death and disability in the world, occurring when a blockage forms in an artery supplying the brain, preventing blood from accessing part of the brain and causing a cascade of events that ultimately cause the death of tissue in the ischaemic region. Reducing the temperature of this tissue has been shown to be beneficial in increasing the treatment window and improving outcomes. Previous modelling studies with vastly idealised geometry have highlighted difficulties in reducing temperatures within the brain using non-invasive means. However by using a model with 1-dimensional vasculature embedded in a 3-dimensional porous tissue, the geometry of the brain and the ischaemic region can be accurately captured. In this work, we simulate a stroke by obstructing a selected vessel in the arterial tree, allowing for varying degrees of severity. By seeking solutions to the mass, momentum and energy equations, we demonstrate that cooling via the scalp has the potential to provide a useful reduction, around 0.5°C, in temperature within the affected area of the brain. The degree of cooling achievable is dependent on the location of the stroke.

1EPSRC and EU-RISE-ThermSMART

8:26AM H18.00003 Flow in Porous Media as a Model for Intramural Periarterial Drainage from the Brain.1, KETAKI JOSHI, Department of Mechanical Engineering, State University of New York at Binghamton, USA, J. DAVID SCHAEFFER, Institute for Justice and Well-Being, State University of New York at Binghamton, USA, PAUL CHIAROT, PETER HUANG, Department of Mechanical Engineering, State University of New York at Binghamton, USA — Accumulation of beta-amyloid proteins in the vasculature of the brain is a characteristic of Alzheimer’s disease. One of the pathways to clear these proteins out of the brain is through intramural periarterial drainage (IPAD) along artery walls. There is evidence that the direction of this flow is opposite to that of the arterial blood flow. The periarterial space where IPAD takes place is mostly made up of smooth muscle cells and extracellular matrix. Deformation of the arterial wall boundaries are theorized to drive IPAD. In our earlier work, we reported on a hydrodynamic mechanism for reverse flow through the artery wall that was driven by forward propagating and reflected waves along the boundaries of an open conduit. In our current work, we have expanded our model by incorporating porous media inside the flow channel. We analyzed images of artery deformations in the brains of anesthetized mice provided by Cornell University (N. Nishimura). The quantified boundary deformations of the cerebral arteries are used in numerical simulations to predict the magnitude of the induced IPAD through the porous periarterial space. The role of geometry (i.e. porosity) and permeability on the fluid transport is determined.

1This project is funded by the National Institute on Aging (NIA/NIH).

8:39AM H18.00004 Computational Study of Hemodynamic Nature in Patient-Specific Cerebrovasculature with Lenticulostriate Artery under ICA Stenosis Conditions, TAEHAK KANG, Department of Mechanical Engineering, Chung Ang University, DEBANJAN MUKHERJEE, Department of Mechanical Engineering, University of Colorado Boulder, JEONG-MIN KIM, KWANG-YEOL PARK, Department of Neurology, College of Medicine, Chung Ang University, JAYOUNG RYU, Department of Mechanical Engineering, Chung Ang University — Lenticulostriate arteries play important role in cerebral circulation, especially in basal ganglia region. Deeper understanding of hemodynamic nature of LSA would grant more effective surgical or endovascular treatments. This is the first study to investigate hemodynamic changes in small cerebral artery, namely lenticulostriate artery (LSA), coupled with complete three-dimensional cerebral vasculature under carotid (ICA) stenosis conditions. For patient-specific study, patient CTA data with complete cerebral anatomy is obtained from STOP Stroke database, and computational reconstruction is carried out using open-source package, SimVascular. Petrov-Galerkin formulation is used to solve the fluid domain with pulsatile inflow and lumped three-element Windkessel outlet conditions. NASCET-based stenosis cases are applied to internal carotid artery for all patient-specific models, and their three-dimensional hemodynamic structure is visualized for comparison. We present the results in key hemodynamic properties and indexes such as VFR, MAP, and OSI. As a result, significant hemodynamic changes are shown in LSA and its cerebrovasculature under progressive ICA stenosis conditions.

8:52AM H18.00005 Cerebral Vascular Super Resolution Imaging and Blood Flow Measurement Using Ultrasound Enhanced Particle Tracking Velocimetry, ZENG ZHANG, JOSEPH KATZ, The Johns Hopkins University, MISUN HWANG, Childrens Hospital of Philadelphia, TODD KILBACH, Children’s Hospital of Philadelphia, PERELMAN School of Medicine, University of Pennsylvania — Hypoxic ischemic encephalopathy is a major cause of neonatal death. It is usually initiated with ischemia and hypoxia followed by blood reperfusion leading to brain damage. While conventional MRI gives hemodynamic information, it cannot be used for cerebral histology, which is vital for understanding the mechanism of reperfusion, and developing appropriate therapies. Therefore, contrast enhanced ultrasound particle tracking velocimetry (echo-PTV) is utilized to generate maps of the cerebral vasculature and blood flow velocity simultaneously. The microbubbles are injected intravenously, and the raw ultrasound images are collected using clinical systems. Blind deconvolution, local background removal, and modified histogram equalization are applied to enhance the image and map the bubble spatial distribution. Subsequently, particle tracking is initiated using cross-correlation and updated with a Kalman filter to calculate the velocity field. The microvascular map is generated by a superposition of the bubble trajectories that can be tracked for over 5 frames. The larger vessels are represented by integrating the enhanced images over the acquisition time. Results show 27% and 48% decrease in blood velocity and micro-vessels number with flow, respectively, 3 hours after ischemia.
9:05AM  H18.00006 Hemodynamics of cerebral aneurysms as cavity flow on a curved vessel. Effect of inertia, curvature, pulsatility and treatment with flow diverting stents

FANETTE CHASSAGNE, MICHAEL C. BARBOUR, VENKAT K. CHIVUKULA, LAUREL M. MARSH, MICHAEL R. LEVITT, ALBERTO ALISEDA, University of Washington, Seattle, WA, USA — Flow in cerebral aneurysms can be described as flow in a cavity on a curved vessel, with the effects of inertia, curvature and pulsatility providing the conditions under which flow separates on the leading edge of the cavity and recirculate inside the sac against the parent vessel flow, for part or all of the cardiac cycle. This strong coherent vortex in the aneurysm can provide a quasi-dissipative metric to explain the persistence of aneurysmal flow after treatment with flow diverting stents (FDS). These high porosity meshes lower the flowrate into the sac promoting the formation of a thrombus. The aim of this study is to investigate the flow pre- and post-treatment with FDS, to understand its variation with flow conditions and aneurysm geometry. Flow inside cerebral aneurysm phantoms with varying parent vessel curvature, neck size and sac aspect ratio is measured with PIV. The hemodynamics in the sac pre- and post FDS treatment are characterized for different waveforms. The results show parent vessel curvature and Dean number decreases the effect of the FDS treatment. This is due to a combination of changes in parent vessel hemodynamics at the aneurysm leading edge combined with the parent vessel curvature impact on stent porosity.

1Funded by: National Institute of Neurological Disorders and Stroke (NIH-NINDS R015088007), National Science Foundation (NSF CBET-0748133)

9:18AM  H18.00007 Coils vs stents: CFD investigation of the mechanisms of healing for two endovascular therapies in cerebral aneurysms

MICHAEL BARBOUR, LAUREL MARSH, Univ of Washington, FANETTE CHASSAGNE, University of Washington, VENKAT KESHAV CHIVUKULA, Univ of Washington, CORY KELLY, SAM LEVY, MICHAEL LEVITT, LOUIS KIM, Univ of Washington Medical Center, ALBERTO ALISEDA, Univ of Washington — Cerebral aneurysm treatment seeks to avoid the risk of rupture by excluding the aneurysmal sac from the circulation. Coiling and stenting are two forms of minimally invasive endovascular treatment that, while increasingly popular, present a non-negligible risk of incomplete embolization of the aneurysm. The placement of flow-diverting stents (FDS) or coils leads to aggregation of activated platelet and thrombus formation. A stable thrombus that fully occludes the aneurysm marks a successful treatment as it excludes the aneurysmal sac from hemodynamic stresses and minimizes the risk of rupture. There is currently no accurate method to predict the outcome of either endovascular therapy. We present in silico studies that show the hemodynamics changes, pre- to post-treatment, for both forms of treatment trend in opposite directions. We consider the treatment types separately to find certain changes in Eulerian variables are statistically different between the successful and unsuccessful treatments. The successfully coiled aneurysm cases were marked by a higher overall increase in the neck plane shear than the failed cases. While one of the most significant metrics for predicting a successful FDS treatment is the change in flowrate entering the aneurysm during systole.

1Funded by: National Institute of Neurological Disorders and Stroke (NIH-NINDS R015088007), National Science Foundation (NSF CBET-0748133), American Heart Association Postdoctoral Fellowship (16POST30520004)

9:31AM  H18.00008 Coils vs stents: Lagrangian-tracking investigation of the mechanisms of healing for endovascular therapies in cerebral aneurysms

MICHAEL MARSH, MICHAEL BARBOUR, FANETTE CHASSAGNE, VENKAT KESHAV CHIVUKULA, University of Washington, CORY KELLY, Univ of Washington Medical Center, SAM LEVY, University of Washington Medical Center, Stroke and Applied Neuroscience Center, MICHEAL LEVITT, LOUIS KIM, Univ of Washington — In this study we investigate the stability of aneurysm flow post-treatment with coils and stents by using a Lagrangian tracking strategy. The flowfield at the leading edge of each aneurysm is measured with particle image velocimetry (PIV) after creating a carotid bifurcation phantom and testing endovascularly treated cerebral aneurysms by means of Lagrangian tracking of platelets in CFD simulations of blood flow in patient-specific models. Flow-diverting stents (FDS) or coils can lead to aggregation of activated platelets and formation of a stable thrombus that fully occludes the aneurysm. However, there is currently no accurate method to predict the outcome of such endovascular therapy. We propose a series of Lagrangian metrics, based on trajectories of tracer particles introduced in the intracerebral vasculature, to understand the hemodynamics conditions that lead to stable thrombus formation as opposed to conditions that lead to incomplete embolization. We consider the two forms of therapies separately, using platelet residence time and shear history to complement Eulerian metrics that have traditionally been used to understand hemodynamics in intracranial aneurysms: Wall Shear Stress, WSS gradient, flow rate through the neck. We develop accurate predictors of outcomes for each endovascular treatment, using a large population of >50 patients to validate them. Your abstract body.

9:44AM  H18.00009 Difference in Hemodynamic Parameters Between Cerebral Aneurysms Models with Truncated Parent Arteries and with Parent Arteries Extending to the Carotid Siphon

YUYA UCHIYAMA, Tokyo University of Science, HIROYUKI TAKAO, The Jikei University of Medicine, SOICHIRO FUJIMURA, HIROSHI OHNO, Tokyo University of Science, FELICITAS DETMER, SARA HADAD, SETAREH SALIMI, FERNANDO MUT, George Mason University, KOJI FUKUDOME, Tokyo University of Science, JUAN CEBRAL, George Mason University, YUICHI MURAYAMA, The Jikei University of Medicine, MAKOTO YAMAMOTO, Tokyo University of Science — Recently, CFD simulations are used to study cerebral aneurysm pathologies. In the human anterior circulation, the internal carotid artery (ICA) has a particular bent section called carotid siphon. However, vessels distal and proximal of aneurysms are often truncated to perform simulations although they can have an effect on blood flow behavior. In this study, we compared hemodynamic parameters in models with and without the siphon. We selected 21 aneurysms from in vivo data using CFD. For each aneurysm (longer model) and the corresponding truncated model, we performed pulsatile blood flow simulations in all the cases, and then calculated and compared 20 typical aneurysm hemodynamic parameters in each model. Results showed that some hemodynamic parameters were different between the two models. Particularly, WSS was on average 60.6% smaller in shorter models compared to longer models. In conclusion, the inclusion of the carotid siphon to the model is important to represent the hemodynamic environment within the aneurysm.

9:57AM  H18.00010 Quantifying unstable flows in sidewall aneurysms at internal carotid aneurysms

TRUNG LE, North Dakota State University, ELIZABETH EIDENSCHINK, ALEXANDER DROFA, Sanford Health — Complex, unstable flow has been linked to aneurysm growth and rupture. Our previous works (Le et al., J. Biomech. Engr., 2010 and Le et al., Annals Biomedical Eng., 2013) have shown a possible transition from the stable mode (cavity) to the unstable mode (vortex ring). We have proposed the use of a novel index called the Enstrophy Number to characterize this transition (Le et al., 2013). However, the quantification of such a transition is lacking. Currently, there has been no efforts in quantifying unstable flows in intracranial aneurysms, which is essential in stratifying not rupture risks, but also to guide clinical management. In this work, the aneurysm geometries from six patients at Sanford Health, North Dakota are reconstructed from Magnetic Resonance Angiogram and Digital Subtraction Angiogram data. Using our in-house CFD code (Virtual Flow Simulator), high-resolution flow data is obtained. In this work, we propose the use of enstrophy mapping as a potential index to stratify aneurysm flows. We show that this type of mapping is able to differentiate stable from unstable modes in patient-specific anatomic. We also provide evidence on the link of Aneurysm Number and enstrophy mapping.

1This work is supported via the start-up funding of Trung Bao Le at the Department of Civil and Environmental Engineering- North Dakota State University and a grant NSF ND EPSCOR #FAR0030612
**10:10AM H18.00011 ABSTRACT WITHDRAWN**

**10:23AM H18.00012 Patient-Specific Computational Modeling of Cerebrovascular AVM-related Aneurysms**

Kimberly A. Stevens, School of Mechanical Engineering, Purdue University; Aaron A. Cohen-Gadol, Goodman Campbell Brain and Spine, Department of Neurological Surgery, Indiana University School of Medicine; Ivan C. Christov, School of Mechanical Engineering, Purdue University; Vitaliy L. Rayz, Weldon School of Biomedical Engineering, Purdue University

An arteriovenous malformation (AVM) is a pathological condition where an abnormal tangle of vessels connects arteries to veins, bypassing the capillary bed. The decreased flow resistance can lead to increased flow in the vessels feeding the AVM, thought to be responsible for the pathogenesis of flow-related aneurysms. To investigate the relationship between AVMs and flow-related aneurysms, we created image-based computational fluid dynamic models of five flow-related cerebral aneurysms coupled with reduced-order lumped-parameter network models representing the AVMs. Vascular geometries for computational models are generated from CT data obtained from IU School of Medicine, and the blood flow within the geometry is computed using an open-source modeling platform, SimVascular. Once created, the model boundary conditions are modified to simulate AVM treatment and predict postoperative flow conditions. Calculated hemodynamic parameters known to influence arterial wall remodeling and intra-luminal thrombus deposition, such as the wall shear stress and oscillatory shear index, are compared between pre- and post-operative flow scenarios. Results indicate that hyperdynamic flow upstream of the AVM leads to elevated wall shear stress, which can be mitigated following AVM treatment.

The research was funded in part by the Lillian Gilbreth Postdoctoral Fellowship.

**Monday, November 25, 2019 8:00AM - 10:36AM**

**Session H19 CFD: Advanced Turbulence Simulations**

Krishnan Mahesh, University of Minnesota, Twin Cities

**8:00AM H19.00001 Turbulent channel flow at \( R e_x = 10000 \)**

Sergio Hoyas, Universitat Politècnica de Valencia; Martin Oberlack, Stefanie Kraheberger, TU Darmstadt; Francisco Alcántara-Avilá, Universitat Politècnica de Valencia

A new simulation of a turbulent channel flow was conducted up to the limit of \( R e_x = 10000 \). The domain size is \( 2\pi h \times 2h \times \pi h \). This domain is thought to be large enough to accurately compute one point statistics of the flow. The simulation has been carried out on 2048 SuperMUC phase II cores, at a mesh of \((6144, 2101, 6144) \approx 8 \times 10^{10} \) grid points. A database with approximately 100 TB has already been created, which will be analyzed further at a later stage. As it was expected, a long logarithmic layer exists with \( k \approx 0.40 \) and extending from \( y^+ \approx 70 \) to \( y^+ \approx 2000 \). The first maximum of the indicator function is not growing anymore and remains constant. A first analysis of the intensities shows that the near wall peaks of \( u' \) and \( w' \) are still growing with \( R e_x \). The possible secondary maximum of \( u' \) is barely present. New scaling laws of \( U' \) and \( u' \) based on symmetry theory will be also shown.

**8:13AM H19.00002 Wall-resolved LES and RANS/LES hybrid analyses in turbulent heat exchanger with modified oblique wavy walls**

Kenichi Morimoto, Shu-Qun Jin, Junyu Chen, The University of Tokyo

High-performance turbulent heat exchangers play a key role in diversified energy systems/devices. In our recent work, double-pipe turbulent heat exchangers with V-shaped oblique wavy walls, with which the ratio of the heat transfer to the pressure loss is much larger than that for conventional techniques, have been proposed. The present study aims to clarify the detailed mechanism of the turbulent heat transfer enhancement, and to explore a practical numerical approach to deal with turbulence anisotropy and unsteady flow separation at high Reynolds number condition. Here we perform wall-resolved large eddy simulations based on dynamic sigma model in which a dynamic procedure is originally applied to both the velocity and thermal fields. Also we perform improved delayed detached eddy simulation (IDDES) with k-ε SST model as a near-wall model. It is shown that, with the present wall undulations, large-scale vortical structures are embedded in a steady manner inside the turbulent boundary layer, leading to remarkable enhancement of the heat transfer performance. The global and local quantities as well as wall shear stress and local wall heat flux are compared between RANS, LES and hybrid approaches. The feasibility of RANS/LES hybrid approach is explored for further shape-optimization study.

**8:26AM H19.00003 Learning about High Schmidt scalar mixing in turbulent round jets from 3D periodic box simulations**

Guillaume Blanquart, Kyupaeeck Jeff Rah, California Institute of Technology

Jet-centerline (JC) forcing technique was developed to create velocity and scalar fields of turbulent round jets in triply periodic box. This forcing method is here utilized to simulate high Schmidt number passive scalars mixing. A series of DNS have been performed to investigate the scalar field statistics of turbulent round jets with Schmidt number \( 10^4 \). This forcing method is here utilized to simulate high Schmidt number passive scalars mixing. A series of DNS have been performed to investigate the scalar field statistics of turbulent round jets. The domain size is \( 8 \times \pi h \) \times \( 8 \times \pi h \) \times \( 8 \times \pi h \). The first maximum of the indicator function is not growing anymore and remains constant. A first analysis of the intensities shows that the near wall peaks of \( u' \) and \( w' \) are still growing with \( R e_x \). The possible secondary maximum of \( u' \) is barely present. New scaling laws of \( U' \) and \( u' \) based on symmetry theory will be also shown.
8:39AM H19.00004 Data-driven predictions of root mean square pressure fluctuations from RANS simulations\textsuperscript{1} , GIACOMO LAMBERTI, CATHERINE GORLE, Stanford University — Wind engineering applications often employ computationally efficient Reynolds-averaged Navier-Stokes (RANS) simulations, even if the turbulence models can compromise the accuracy of the prediction. Furthermore, the estimation of turbulent quantities such as fluctuating pressure loads requires additional, potentially inaccurate, models. For example, empirical models to estimate the root mean square (rms) pressure fluctuations from of the mean pressure, turbulent kinetic energy and velocity, are known to produce incorrect results on the lateral facades of tall buildings. To address this problem, we propose a data-driven approach to determine the best functional form that relates high-fidelity data of rms pressure fluctuations to time-averaged variables predicted by RANS. The high-fidelity data is obtained from large-eddy simulations or wind tunnel experiments of a tall building at different wind directions. We perform RANS simulations for the same conditions and construct features from the resulting time-averaged quantities. Then, we employ machine learning to relate these features to the available training data. We investigate the ability of the data-driven model to provide predictions for wind directions or regions of the building that were not included in the training data.

\textsuperscript{1}This research is funded by NSF Grant Number 1635137.

8:52AM H19.00005 Area of Scalar Isosurfaces in Homogeneous Isotropic Turbulence\textsuperscript{2} , KEDAR PRASHANT SHETE, STEPHEN DE BRUYN KOPS, University of Massachusetts Amherst — A fundamental effect of fluid turbulence is turbulent mixing, which results in the stretching and wrinkling of scalar isosurfaces. Thus, the area of isosurfaces is of interest in understanding turbulence in general with specific applications in, e.g., combustion and the identification of turbulent/non-turbulent interfaces. We report measurements of isosurface areas in 28 direct numerical simulations (DNSs) of homogeneous isotropic turbulence with a mean scalar gradient resolved on up to 14256\textsuperscript{3} grid points with Taylor Reynolds number $Re$ ranging from 24 to 633 and Schmidt number $Sc$ ranging from 0.1 to 7. The continuous equation we evaluate converges exactly to the area in the limit of zero layer thickness. We demonstrate a method for numerically integrating this equation that, for a test case with an analytical solution, converges linearly towards the exact solution with decreasing layer width. By applying the technique to DNS data and testing for convergence with resolution of the simulations, we verify the resolution requirements for DNS that have been recently observed. We conclude that isosurface areas scale with the square root of the Taylor Péclet number $Pe$ between approximately 50 and 4429. No independent effect of either $Re$ or $Sc$ were observed.

\textsuperscript{2}Frontier Project FP-CFD-FY14-007

9:05AM H19.00006 A study of locking phenomenon of elliptical particle in shear flow with DNS . ZHIZHONG DING, Louisiana State University; Shell Company, CHENGUANG ZHANG, MIT, SHASHANK TIWARI, Louisiana State University, JYESHTHARAJ JOSHI\textsuperscript{1}, Institute of Chemical Technology, Mumbai, KRISHNASWAMY NANDAKUMAR, Louisiana State University, INSTITUTE OF CHEMICAL TECHNOLOGY, MUMBAI COLLABORATION, LOUISIANA STATE UNIVERSITY COLLABORATION — The understanding of the dynamics of arbitrary shaped particle(s) under shear flow is of great importance in design and scale-up Chemical Engineering equipment. In this study, we have carried out a series of numerical experiments on an elliptical cylinder particle, subjected to a 2D shear flow for moderate shear-based Reynolds number. The simulations have been performed using an in-house code Signed Distance Field - Immersed Boundary Method (SDFIBM) (Zhang et al., 2018). The degrees of freedom of the particle is limited and thus allows the particle to rotate only about a fixed position in space. Any translational and vertical movements are strictly limited to phase out the noises. This work demonstrates that there are two stages for elliptical cylinder particle in shear flow: the periodical rotating stage and stationary stage, divided by a critical Reynolds number. We further extend our investigation to include the effects of changing particle aspect ratio, flow confinement size and distance of particle from the wall. Frequency domain analysis have been carried out on the data to develop a better understanding of the locking phenomenon. The effect of a second ellipse in the vicinity of primary particle and its impact on the locking phenomenon has also been included.

\textsuperscript{1}One more affiliation he has: Homi Bhabha national institute, Mumbai

9:18AM H19.00007 Direct Numerical Simulation in the Sub-critical and Supercritical regimes for Flow past a Stationary Sphere , SHASHANK S TIWARI\textsuperscript{1}, Institute of Chemical Technology, Mumbai, SHIVKUMAR BALE, University of Pittsburgh at Johnstown, ZHIZHONG DING, Louisiana State University; Shell Company, ASHWIN W. PATWARDHAN, Institute of Chemical Technology, Mumbai, KRISHNASWAMY NANDAKUMAR, Louisiana State University, JYESHTHARAJ B. JOSHI\textsuperscript{2}, Institute of Chemical Technology, Mumbai — The physical understanding of separating flows which exhibit critical phenomena under various flow conditions, have helped in designing various drag-reduction devices, turbulence generators, controllers, etc. Direct Numerical Simulation (DNS) of such separating flows helps to identify the flow structures and decipher the corresponding effects they have on the resulting forces. DNS being computationally intensive, the investigations for flow past a sphere has been limited to $Re \approx 10,000$. In this study, we test the capability of OpenFOAM in performing fully resolved DNS for $1000 < Re < 10^5$ Appropriate time and length scales have been used to adequately resolve the boundary layer. Extensive simulations were performed to test, optimize and set a benchmarking case for the domain size, mesh size, time-step and discretization schemes required for performing such computationally intensive simulations on a scalable parallel platform. Simulations were run till a statistically converged solution was obtained. The drag coefficients and pressure coefficients from the simulations were compared against experimental results available in literature and were found to be in good agreement.

\textsuperscript{1}Louisiana State University

\textsuperscript{2}Homi Bhabha National Institute

9:31AM H19.00008 Large-Eddy Simulation of Propeller Crackback Using an Unstructured Overset Grid Method\textsuperscript{3} , THOMAS KROLL, WYATT HÖRNE, KRISHNAN MAHESH, University of Minnesota — We discuss the application of a novel unstructured overset grid methodology to compute the flow around a marine propeller. We consider the crackback mode of operation where the propeller rotates in the reverse direction while the vessel moves in the forward direction yielding highly unsteady loads. The numerical algorithm is that developed by Horne and Mahesh [J. Comput. Phys. (2019) 376:585-596] and addresses two significant challenges posed by the overset methodology - discrete conservation and scaling. The simulations consider open and ducted propellers. The results are compared to available experimental data and previous LES studies. Details of the flow field are discussed.

\textsuperscript{3}This work is supported by the Office of Naval Research (ONR).
In particular, we study the error in the transition location and the mean skin-friction due to the different coarse ($h$) and medium ($p$) meshes. The flow problem is simulated using a cylindrical grid composed of about 1.7 billion nodes. Earlier computations of the isolated propeller (open-water condition) demonstrated the accuracy of the overall methodology, via comparisons with both dynamometric measurements and Particle Imaging Velocimetry (PIV) visualizations. Three incidence conditions of the rudder are considered, corresponding to $0^\circ$, $10^\circ$ and $20^\circ$. Comparisons with the results in open-water conditions demonstrate that in the near wake the topology of the typical coherent structures shed by submarine propellers (tip and hub vortices) is not modified by the presence of the upstream hydrofoil at small incidence angles. In contrast, levels of turbulence within the wake are dramatically increased in the last configuration, featuring the rudder at $20^\circ$ of incidence. This substantial change is triggered by the separation occurring on the suction side of the hydrofoil, leading to a significant perturbation of the inflow conditions of the propeller.

We would like to acknowledge the support from the AFOSR under grant FA9550-16-1-0128, and US Army Research Office under grant W911NF-15-1-0505

1AP supported by HOLISHIP (EU H2020), grant n.689074; EB partially supported by ONR Grant N00014-18-1-2671. HPC resources provided by PRACE (Marconi-KNL at CINECA), project n. 2016163889.

10:10AM H19.00011 On the effect of wake passing on a low pressure turbine cascade using spectral/hp element methods, ANDEA CASSINELLI, Imperial College London, PAOLO ADAMI, Rolls-Royce plc., FRANCESCO MONTOMOLI, SPENCER J SHERWIN, Imperial College London — The developing and future interaction of HPC, high fidelity CFD and high order unstructured grid algorithms has the potential to allow for simulation-based research, analysis and design capability. Previous work focused on developing guidelines to leverage the use of high order spectral/hp element methods as a virtual cascade for turbomachinery applications. Building on the knowledge previously reported, we analyze a representative industrial low pressure turbine cascade subject to wake passing interactions at moderate Reynolds number, adopting the incompressible Navier-Stokes solver implemented in the Nektar++ software framework. The rotor-stator interaction is modelled by imposing appropriate Dirichlet boundary conditions. The impact of flow coefficient and reduced frequency is studied in conjunction with the Reynolds sensitivity. The analysis focuses in detail on the dynamics of the separation bubble on the suction surface looking at mean flow properties and turbulence kinetic energy budgets, comparing the main findings with experimental data.

1We would like to thank EPSRC for the computational time made available on the UK supercomputing facility ARCHER via the UK Turbulence Consortium under grant EP/R029326/1, as well as the Research Computing Service at Imperial College London. We would also like to thank Rolls-Royce plc. for support and permission to publish this work.

10:23AM H19.00012 Numerical investigation of the flow in gas turbine blade trailing edge internal cooling passages, JAEHYUN RYU, WONTAE HWANG, Seoul National University — Gas turbine blades operate at very high temperatures, often beyond material limits. Internal and external cooling enable the blade to survive these extreme temperatures. The trailing edge of the blade is designed to be sharp for high aerodynamic efficiency, but this makes internal convective cooling poor at the corner. The effect of ribs was assessed in a right triangle channel containing a sharp corner, representing a simplified trailing edge. First, 45 angled ribs on the pressure and suction side walls were investigated. The Reynolds-averaged Navier-Stokes (RANS) results show fairly good agreement with previous results from magnetic resonance velocimetry (MRV) and large eddy simulation (LES). Different turbulence models were assessed, and the baseline explicit algebraic Reynolds stress model (BSL EARSM) was adequate in capturing the flow structure. Next, we optimized rib geometry via design of experiments (DOE), by changing the rib height to channel height ratio and rib angle. 3-level full factorial design (FFD) was used to determine the DOE points. Streamwise flow velocity at the sharp corner and friction factor were set as objective functions. The corner flow velocity increases as the rib is angled less toward the flow, at the penalty of an increase in friction factor.

Monday, November 25, 2019 8:00AM - 10:10AM – Session H20 Geophysical Fluid Dynamics Cryosphere 602 Claudia Cenedese, Woods Hole Oceanographic Institution

8:00AM H20.00001 Modeling the formation of wind-driven coastal polynyas, LAILAI ZHU, Princeton University, BHARGAV RALLABANDI, University of California, Riverside, MICHAEL WINTON, NOAA/Geophysical Fluid Dynamics Laboratory, HOWARD STONE, Princeton University — Polynyas are persistent, recurrent regions of open sea water surrounded by sea ice and/or land in the polar zones. We conduct a combined theoretical and numerical study on the dynamics and thermodynamics of the formation of a wind-driven latent-heat polynya near a coastline with and without curvature. In the limit of weak ice internal pressure, we propose a one-dimensional, continuous, mass- and momentum-conserving theory characterizing the offshore distribution of the ice velocity and the spatial-temporal evolution of the ice concentration. Using an open-source solver based on finite-element method, we simulate this process considering the rheological properties of the ice. The theoretical and numerical results agree well with each other in most cases, and the obtained steady-state polynya width qualitatively matches the corresponding climate data.
JUMPING FROST

8:26AM H20.00003 Lubricated gravity currents of power–law fluids1, AYALA GYLLENBERG, ROIY SAYAG, Ben Gurion University — The discharge of ice from power–law fluids into the ocean has the potential to induce climate change and a catastrophic rise in sea level. Significant ice discharge can occur via ice streams, bands of fast–moving ice lubricated by a mixture of water and clay. We present a fluid–dynamical model for such lubricated gravity currents that describes the axisymmetric spreading of a viscous, non–Newtonian fluid, such as ice, under thinning and melting. Both fluids are discharged at the origin at a time–dependent flux of a general power–law form. We investigate the model solutions by combined analytical and numerical methods. We find that the model admits self–similar solutions only in specific cases, such as when the top fluid is Newtonian or when the fluids are discharged at a certain time–dependent flux. When the flux is constant, similarity solutions are present when the viscosity of one fluid is much greater than that of the other fluid, as in the case of an ice sheet lubricated by water. In that case we find that the front of the lubricating fluid outstrips that of the power–law fluid, a phenomenon that has been observed in laboratory experiments.

1German-Israeli Foundation 1-2404-301.8/2015

8:39AM H20.00004 Lubricated gravity currents of strain–rate softening fluids, and the formation of ice-streams1, ROIY SAYAG, Ben Gurion University, PRAMODA KUMAR, Ashuka University, SHAHAR ZURI, Technion — Ice–sheet instabilities are believed to be closely associated with the flow of ice streams, bands of fast–flowing ice that carry most of the mass flux towards the edge of the ice sheet, where it ultimately melts or calves. The formation and evolution of ice streams is believed to be controlled by a complex hydrological network under the ice that sets the dynamic boundary conditions on the base of the ice. Fluid mechanically, such a system can be modelled as a viscous gravity current of strain–rate softening fluid lubricated by a relatively inviscid and denser Newtonian fluid. We present an experimental study of such flows that were discharged axially and radially at a constant rate. We investigate the evolution of the fronts of the viscous and the lubricating fluids, and how and why they form depend on the manifold dimensionless groups of the flow. We find two distinguished classes of patterns. In the first class, both fronts remain axisymmetric throughout the flow. In the second class, the initially axisymmetric front of the lubricating fluid evolves into a set of localised fingers, which drive the initially axisymmetric flow of the viscous fluid to develop patterns that are reminiscent to ice streams.

1German-Israeli Foundation 1-2404-301.8/2015

8:52AM H20.00005 Ice-Shelf Rippling From Temporally Varying Melt and Ice Flow, ANDREW WELLS, CHRIS MACMACKIN, University of Oxford — The Antarctic and Greenland ice sheets often discharge into the ocean through floating ice shelves. Ice-shelf geometry is controlled by the coupling of a viscous ice flow with basal melt driven by buoyant meltwater plumes rising through the warm and salty ocean along the sloping ice-shelf base. Motivated by recent observations of the spatial patterns of ice-shelf thickness, we assess the response of an ice shelf to a range of temporally-periodic forcing conditions. We model the depth-integrated viscous extensional flow of an ice shelf downstream of the grounding line, where the grounded ice transitions to the floating shelf. Melting is determined from a meltwater plume model with imposed subglacial discharge. We use a linear perturbation analysis to explore the sensitivity of the ice shelf geometry to periodic variation of the freshwater source flux for the meltwater plume, ice flux across the grounding line, and coupled variation of ice flux and grounding line position. Such periodic variation leads to propagating ripple-like features on the ice shelf base, with greatest sensitivity to varying ice influx with coincident grounding line motion. We analyse the mechanisms controlling ripple amplitude, and discuss potential implications for formation of ice-shelf basal terraces.

9:05AM H20.00006 Relaxation of ice-sheet uplift on a porous bed, CHING-YAO LAI, Columbia University, DANIELLE L. CHASE, Princeton University, LAURA A. STEVENS, TIMOTHY T. CREYTS, Columbia University, HOWARD A. STONE, Princeton University — When surface meltwater reaches the base of an ice-sheet with high flow rate, it peels apart the interface between the ice and bed, forming a water-filled blister and inducing ice-sheet uplift. The uplift relaxes after water injection, with a wide range of relaxation times (Geodetic observations show relaxation occurs over hours to days). Improved characterization of water flow at the base of ice sheets is vital for predicting ice-sheet dynamics and mass loss when the bed is lubricated. Although properties of the lubricated porous sheet beneath the ice impact ice sliding, the permeability of the porous sheet has not been measured directly and remains unknown. Here, we show that the elastic relaxation of a water-filled blister above a porous substrate can be used to probe permeability. Combining field data, a mathematical model, and laboratory experiments, we show that the blister height decreases exponentially with time as the water in the blister permeates through the porous sheet. We find that the relaxation dynamics obeys a universal law with a time scale involving the thickness and permeability of the porous sheet, water viscosity, and the Young’s modulus of ice. We show that the range of observed relaxation times can be explained by the evolving permeability of the porous sheet. Our reduced-order model better characterizes the evolution of bed permeability based on surface observations.

9:18AM H20.00007 Direct numerical simulations of ice melting in a turbulent ocean1, LOUIS-ALEXANDRE COUSTON, British Antarctic Survey — In this talk I will present preliminary results of direct numerical simulations of ice melting in a stratified turbulent ocean. The model solves the evolution of the turbulent fluid phase and of the diffusive solid ice phase, due to melting and freezing, in a fully coupled way. This is done by combining a highly-efficient fully-spectral Direct Numerical Simulation (DNS) code with a novel formulation of the equations for the solid and liquid phases of water based on the phase-field method, which is routinely used in metallurgy. DNS enables turbulent motions to be simulated without approximation, i.e. solving Navier Stokes equations, while the phase-field method allows the ice-ocean interface to be rough and evolve in response to melting. I will present results on the turbulent boundary layer and on the self-generated roughness at the ice-ocean interface for fresh water. The ultimate goal of this work is to propose a new DNS-based parameterization of the melting process at rough ice-ocean boundaries that takes into account the effects of temperature and salt stratification, and flow velocities.

1The project leading to this application has received funding from the European Unions Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No 793450
9:31AM H20.00008 Glacial Squeegee: Elastic Landau-Levich and the Tidal Modulation of Ice Streams, KATARZYNA WARBURTON, DUNCAN HEWITT, JEROME NEUFELD, University of Cambridge — Glacier speed is sensitive to fluctuations in subglacial water pressure. For marine ice sheets, the tidal cycle is linked to the upstream pressure fluctuations as water enters and exits the subglacial environment across the grounding line. We analyse the elastic analogue of the Landau-Levich dip-coating problem, in which a plate (here the earth) is withdrawn from a bath of fluid (the ocean) on whose surface lies a thin elastic sheet (the ice), for arbitrary angle of withdrawal \( \delta \) (basal slope). The flow is controlled by the elasticity number, \( E_l \), which is a measure of the relative importance of viscous and bending stresses, and \( \delta \). The leading order solution for small \( E_l \) is a steady profile in which the thickness of the film deposited on the plate is found. The breakdown of this asymptotic regime in the limit of large \( E_l \) is also discussed. These solutions are interpreted in the context of the tidal grounding line problem, and related to observations from the Rutford Ice Stream.

9:44AM H20.00009 Ice floe dispersion from moderate remote sensing imagery\(^1\), ROSALINDA LOPEZ, University of California Riverside, MONICA M. WILHELMS, Department of Mechanical Engineering, University of California Riverside, WILHELMS LAB TEAM — Sea ice transport directly affects the heat budget and freshwater flux in the Arctic. To understand the Arctic climate system, it is thus important to quantify the dispersion regime of free-drifting sea ice. In this study, we employed an in-house algorithm to automatically process Moderate Resolution Imaging Spectroradiometer satellite images and track ice floes (0 to 65 km) along the eastern coast of Greenland during the spring of 2017. By quantifying the drift fields, the dispersion regime of the ice floes was analyzed to understand the drivers of sea ice transport. Preliminary results show that, as expected, in the presence of a strong shear flow as the East Greenland Current, the absolute dispersion of ice floes grows quadratically in time. Further analysis of the fluctuating component of sea ice velocities also yields an absolute dispersion that grows quadratically in time at short time scales (few days). It was observed, however, that this behavior changes when considering longer time scales as the influence of the underlying eddy field becomes more prominent. We examine the effect of small scale oceanic turbulence on sea ice drift and discuss the feasibility of extending our study to improve our understanding of sea-ice ocean interactions.

\(^1\)Graduate Assistance in Areas of National Need (GAANN) program from the Department of Education

9:57AM H20.00010 Enhanced submarine melting of glaciers due to the effect of sediments on subglacial discharge plumes\(^1\), CLAUDIA CENEDESE, Woods Hole Oceanographic Institution, CRAIG MCCONNOCHIE, University of Canterbury, JIM MCELWAINE, Durham University — Recent studies suggest that marine terminating glacier faces meet at particularly high rates where subglacial discharge plumes are present. These subglacial discharge plumes have high sediment concentrations, as revealed by the surface expression of some of them, yet the impact of suspended sediment on the plume dynamics and the melting has, until now, been overlooked. Recent laboratory experiments and Direct Numerical Simulations (DNS) suggest that the sediment load could have an important and complex effect on the plume dynamics and thus the melting of the ice face. In particular, dilute concentrations of small and dense particles can increase the entrainment of ambient waters into the plume as much as 50\%, leading to a 50\% increase in the melt rate. In these experiments and simulations, the settling velocity of the particles is smaller than and in the opposite direction to the plume load could have an important and complex effect on the plume dynamics and thus the melting of the ice face. In particular, dilute concentrations of small and dense particles can increase the entrainment of ambient waters into the plume as much as 50\%, leading to a 50\% increase in the melt rate. In these experiments and simulations, the settling velocity of the particles is smaller than and in the opposite direction to the plume velocity at all heights and particle inertial clustering, leading to convective instability in the fluid, is thought to be the mechanism responsible for the enhanced entrainment.

\(^1\)NSF OCE-1658079

Monday, November 25, 2019 8:00AM - 10:36AM — Session H21 Drops: Impacts with Solids I

8:00AM H21.00001 Droplet Dynamics Post Oblique Surface Impact\(^1\), VANESSA KERN, CATHY JIN, PAUL STEEN, Cornell University — Oblique droplet impact onto dry surfaces is ubiquitous in industry and nature, yet scarce in the literature. Here we investigate the post-impact dynamics of capillary water droplets obliquely impacting homogeneous chemically-prepared hydrophilic and hydrophobic surfaces. We analyze high-speed images of the impact event. Pre-impact Weber numbers range from 0-16. Impinging angles range from 0-53\(^\circ\). We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones. Post-impact Weber numbers range from 0-16. We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones. Post-impact Weber numbers range from 0-16. We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones. Post-impact Weber numbers range from 0-16. We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones. Post-impact Weber numbers range from 0-16. We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones. Post-impact Weber numbers range from 0-16. We observe that the drop spreads to a greater extent and forms a thinner film on hydrophilic surfaces than on hydrophobic ones.

\(^1\)NSF-1530522

8:13AM H21.00002 Contact-line Fluctuations and the Dynamics of Wetting, JOEL DE CONINCK, University of Mons — Using MD simulations, we have shown previously that the fluctuations about the mean position of the contact line may be interpreted in terms of an overdamped one dimensional Langevin harmonic oscillator of stiffness \( k \) and demonstrated a relationship between the time scale of the fluctuations of the contact line, the time decay of the oscillations, and the contact-line friction. Here, we extend this work to study the fluctuations when the contact lines are moving, yielding dynamic advancing and receding contact angles that differ from their equilibrium values. A steady dynamic state is achieved by moving the plates in opposite directions at constant velocities. Under these conditions, we obtain an identical Langevin expression to that found at equilibrium, but now with the harmonic term centered about the mean contact line and a fluctuating capillary force arising from the fluctuations of the dynamic contact angle around its mean value. During the talk, we will show that the contact-line fluctuations are the same irrespective of whether the contact line is at equilibrium or moving and contain all the information necessary to predict the dynamics of wetting.

8:26AM H21.00003 Contact time of water drop on cylindrical superhydrophobic surfaces\(^1\), CHONGYEOP LEE, JEONGHOON HAN, Kyung Hee University, WONJUNG KIM, Sogang University, CHANGWOOW BAE, Kyung Hee University, DONG WOO LEE, SEUNGWON SHIN, Hongik University, YOUNGSUK NAM, Kyung Hee University — When the water drop is impinged upon a cylindrical superhydrophobic surface with varying diameter, its contact time with the surface decreases over that on the flat superhydrophobic surface. However, the prediction of the contact time remains challenging due to the complex drop spreading dynamics on cylindrical superhydrophobic surface after impact. Here, based on systematic experimental and numerical studies, we develop a scaling relationship for the contact time reduction on the cylindrical superhydrophobic surface. We show that non-dimensionalized contact time can be expressed as a function of a single dimensionless parameter, which is expressed as the combination of the Weber number and the ratio of the cylinder diameter to the drop diameter.

\(^1\)National Research Foundation of Korea - 2017R1A2B4008028, 2019R1A2C2004607, 2017R1D1A1B03028518
gas on splashing at high Weber and Reynolds numbers, project no. BA 4953/3. Gordillo [Phys. Rev. Lett. 113, 024507 (2014)], who proposed that these aerodynamic forces are the responsible for splashing.

The results show that, under our experimental conditions, liquids spread with a maximum advancing contact angle greater than 87 degrees, regardless of the liquid or substrate properties. Our results also show that existing dimensionless groups, i.e. the splashing parameter (K) and the capillary number (Ca), are not appropriate to characterise the splashing behaviour. Finally, we show that the splashing ratio β and the maximum dynamic advancing contact angle, appropriately divides the splashing and no-splashing behaviour.

1 Royal Society (Grants No. URFuRn180016 and No. RGFuEAn18002), the EPSRC (Grant No. EP/P024173/1), the Mexican Energy Ministry (SENER), CONACyT Mexico, and the Welsh Funding Office (WEO)

8:39AM H21.00004 Role of the Dynamic Contact Angle on Splashing1, MIGUEL A. QUIETZERI-SANTIAGO, Queen Mary University of London, KÉNSUKE YOKOI, Cardiff University, ALFONSO A. CASTREJN-PITA, University of Oxford, J. RAFAEL CASTREJN-PITA, Queen Mary University of London — A drop impacting onto a solid dry substrate can, among other several results, splash or spread over the solid surface. The result depends not only on the droplet properties and speed, but on a wide range of parameters. Although many studies have aimed at finding scaling arguments to characterise splashing, the exact combination of parameters and their influence have remained elusive. In this work we perform a systematic study of liquid droplets impacting onto various solid substrates ranging from completely wetting to superhydrophobic. The experimental approach uses high-speed imaging and image analysis to recover the contact angle as a function of the spreading velocity. We show that, under our experimental conditions, liquids spread with a maximum advancing contact angle greater than 87 degrees, regardless of the liquid or substrate properties. Our results also show that existing dimensionless groups, i.e. the splashing parameter (K) and the capillary number (Ca), are not appropriate to characterise the splashing behaviour. Finally, we show that the splashing ratio β and the maximum dynamic advancing contact angle, appropriately divides the splashing and no-splashing behaviour.

8:52AM H21.00005 Impact dynamics of ferrofluid drop on superhydrophobic surface under horizontal magnetic field1, NILAMANI SAHOO, DEVRAJAN SAMANTA, PURBARUN DHAR, Indian Institute of Technology Ropar — In this study, we focus on impact of ellipsoidal or columnar droplet by horizontal magnetic field on a non-wetting substrate and investigates experimentally the effect of Weber number (We) and magnetic Bond number (Bo_m). The orthogonal spreading depends on the magnitude of applied magnetic field (manifested through Bo_m), since the applied field alters the spherical shape of pre-impact drop into either ellipsoidal or columnar drop. The orthogonal spreading induces non- axisymmetric distribution pre-impact inertial energy along the transverse and longitudinal axes of the applied magnetic field, promoting rebound suppression for a fixed We. With increase in Bo_m, the nature of orthogonal spreading is more prominent to break symmetry retraction as observed experimentally. In addition, at higher orthogonal spreading ratio, the shattering of liquid lamella occurs due to nucleation of holes at different locations after post impact during retraction phase. We can suggest that the shattering of liquid lamella is critically dependent on the thickness of the liquid film after post impact and the wettability of the surface.

1 DS and PD would like to thank IIT Ropar for funding the present work (vide grants 9-246/2016/IITRPR/144 & IITRPR/Research/193 respectively)

9:05AM H21.00006 A theory on the spreading of droplets, JOSE GORDILLO, GUILLAUME RIBOUX, ENRIQUE S. QUINTERO, Universidad de Sevilla — Here we provide a self-consistent analytical solution describing the unsteady flow in the slender thin film which is expelled radially outwards when a drop hits a dry solid wall. Thanks to the fact that the fluxes of mass and momentum entering into the toroidal rim bordering the expanding liquid sheet are calculated analytically, we show here that our theoretical results closely follow the experimentally measured time-varying position of the rim with independence of the wetting properties of the substrate. The particularization of the equations describing the rim dynamics at the instant the drop reaches its maximal extension which, in analogy with the case of Savart sheets, is characterized by a value of the local Weber number equal to one, provides an algebraic equation for the maximum spreading radius also in excellent agreement with all the experimental data available in the literature. The self-consistent theory presented here provides us with the time evolution of the thickness and of the velocity of the rim bordering the expanding sheet. We also test these predictions and show that our theory is also able to predict the splash threshold velocity when the substrate is superhydrophobic and also the velocities and the diameters of the droplets ejected.

9:18AM H21.00007 Axisymmetric Lattice Boltzmann Simulation of Droplet Impact on Spherical Surfaces1, XIN YONG, HUSSIN DALGAMONI, Binghamton University — Droplet impact on solid surfaces plays important roles in many engineering applications, in which the ability to exert control over the detailed dynamics is critical. While past studies have established a complete understanding of droplet impact on flat substrates, what we know about the impact dynamics on curved surfaces is limited. This work simulated the normal impact of droplets on spherical surfaces with physical density and viscosity contrasts in the low Weber number regime, in which droplet deformation is assumed to be axisymmetric. We extended our recently developed axisymmetric free-energy lattice Boltzmann method (LBGM) to capture droplet wetting and contact line motion on a curved surface. The conventional staircase approximation of curved boundaries was used to reduce computational cost. Its accuracy was validated by the static wetting simulations. The impact simulations show that surface curvature and wettability significantly affected the spreading and recoiling of droplet. Five impact outcomes were observed, which ranges from deposition to total rebound. An impact phase diagram was constructed and correlated with the total contact time to provide guidelines for surface design for anti-icing applications.

1 This research was partially supported by the National Science Foundation (Award 1538090).

9:31AM H21.00008 Experimental and numerical study of the effect of surrounding gas on splashing at high Weber and Reynolds numbers1, DAVID A. BURZYNSKI, STEPHAN E. BANSMER, Technische Universität Braunschweig — We investigated the influence of the surrounding gas on a droplet impacting a smooth dry glass surface at high Weber and Reynolds numbers. We analyzed experimentally the splashing outcome by measuring the size, velocity, and angle of the secondary droplets and by calculating the total volume ejected. Numerical simulations complement our study by providing detailed information about the flow in the liquid lamella and the surrounding gas. The results show that gas entrainment is not the mechanism responsible for splashing and demonstrate that splashing is influenced first by the density, second by the viscosity, and lastly by the mean free path of the surrounding gas. The simulations are used to estimate the forces acting on the ejected lamella and to compare it with the theory of Ribol & Gordillo [Phys. Rev. Lett. 113, 024507 (2014)], who proposed that these aerodynamic forces are the responsible for splashing.

1 This research was supported by the German Research Foundation (Deutsche Forschungsgemeinschaft, DFG) through financing of the project no. BA 4953/3.
9:44AM H21.00009 Surrounding gas-independent splash at high-velocity drop impact with a projected smooth solid surface\(^1\). MASAo WATANABE, TAKU ASHIDA, KAZUMICHI KOBAyASHI, HIROYUKI FUJII, Hokkaido University, TOSHIYUKI SANADA, Shizuoka University — We study a drop impact on a fast-moving smooth solid plate in a reduced-pressure condition. A water drop of radius \(R = 1.1\) mm was released from a rest needle in a stainless vacuum chamber; then, the free falling drop was brought into collision with a vertically upward flying solid impact plate which was projected by an iron bullet accelerated by a coilgun. The impact plate consisted of a cover glass with a static contact angle of 60° and a surface roughness \(R\) of 2.1 nm, adhered to an acryl plate. The surrounding gas pressure was varied between 1 and 100 kPa, and the impact velocity was varied between 4.2 and 33 m/s. The drop impact on a smooth solid plate with high impact velocity and the subsequent splash was recorded by a high-speed video camera at a frame rate of 1,000,000 fps at pixel resolution of 16.1 \(\mu\)m/pixel. In a reduced-pressure condition, preceding the occurrence of corona splash, fine daughter droplets, possibly smaller than those observed in corona splash, fly extremely fast along the solid surface only for a few microseconds. The experimental results show that this splashing is independent of both the surrounding gas pressure and gas species. We discuss the threshold of the occurrence of this splashing and identify the mechanism for this splashing.

\(^1\)The authors thank Society for the Promotion of Science, JSPS KAKENHI Grant Numbers JP16K06073 and JP17H03168, for supporting this work.

9:57AM H21.00010 Air entrapment under drop impacts on soft solids\(^1\). KENNETH R. LANGLEY, KAUST, ALFONSO A. CASTREJON-PITA, University of Oxford, S. T. THORODDSEn, KAUST — Prior studies have found that soft solids delay the critical velocity at which drops begin to splash upon impact [1]. We investigate the effects of the surface compliance on the air cushioning at the bottom of a liquid drop impacting onto a soft solid and the resulting entrapment of a central bubble using high-speed interferometry at 5 million frames per second and spatial resolution of 1.05 \(\mu\)m/pixel. The soft solid delays the effects of gas compressibility. We also observe extended gliding of the drop as it initially avoids contact with the surface and spreads over a thin layer of air and investigate the threshold velocity for the onset of gliding. Such extended gliding layers have previously been seen for high viscosity drop impacts [2], but not for low viscosity drops. Additionally, we observe the dynamics as the drop spreads near the splashing threshold to observe effects of the compliance on the ejected lamellae. [1] Howland, C. J., Antkowiak, A., Castrejón-Pita, J. R., Howison, S. D., Oliver, J. M., Style, R. W., & Castrejón-Pita, A. A. (2016). Phys Rev Lett, 117 (18), 184502. [2] Langley, K., Li, E. Q., & Thoroddsen, S. T. (2017). J Fluid Mech, 813, 647-666.

\(^1\)This work was funded by King Abdullah University of Science and Technology (KAUST) under grant URF/1/3727-01-01.

10:10AM H21.00011 Characterizing the wettability/non-wettability transition during drop impact\(^1\). HARISH DIXIT, PRAVEEN SHARMA, Indian Institute of Technology Hyderabad — Recent experiments have shown a new bouncing mechanism on smooth surfaces where the drop is supported on a thin cushion of gas beneath it. Using incompressible high-resolution computations, we study drop impact dynamics at moderate speeds of impact. We restrict our attention to parameters where drop undergoes complete rebound and show that five distinct regimes can be identified in a Reynolds-Weber number phase diagram. In a broad sense, a stable gas cushion can be formed for low Reynolds number but for a wide range of Weber numbers. With increasing Reynolds/Stokes number, the gas film undergoes rupture either in an annular ring near the drop periphery or near the center of the drop. Each of the five regimes differs in the shape of the gas film near the impact surface. The exact transition boundary between wettability-independent and wettability-dependent bouncing is difficult to determine, but the simulations are found to be in good agreement with known scaling laws for initial deformation height, maximum spreading radius and minimum thickness of the gas film attained. Access to velocity and surface energy scaling laws is difficult to determine, but the simulations are found to be in good agreement with known scaling laws for initial deformation height, maximum spreading radius and minimum thickness of the gas film attained. Access to velocity and surface energy scaling laws is difficult to determine, but the simulations are found to be in good agreement with known scaling laws for initial deformation height, maximum spreading radius and minimum thickness of the gas film attained.

\(^1\)SERB grant ECR/2015/000086. Dept. of Science & Technology

10:23AM H21.00012 Some additional considerations on the splashing of droplets. GUILLAUME RIBOUX, JOSE GORDILLO, Universidad de Sevilla — When a drop of a low viscosity liquid impacts against a smooth solid substrate at a velocity \(V\), a liquid sheet of thickness very small compared to the drop radius is expelled tangentially to the substrate at high velocity \(V^*\). If the impact velocity is such that \(V > V^*\) with \(V^*\) the critical velocity for splashing, the edge of the expanding liquid sheet lifts off from the wall as a consequence of the gas lubrication force at the wedge region created between the advancing liquid front and the substrate. In the present talk, we show that the magnitude of the gas lubrication force is limited by the values of the slip lengths at the gas-liquid interface and at the solid. We demonstrate that the splashing regime changes depending on the value of the ratio of the slip lengths, a fact explaining the spreading-splashing-spreading-splashing transition for a reduced value of the surrounding gas pressure as the drop impact velocity increases. We also provide an expression for \(V^*\) as a function of the inclination angle of the substrate, the drop radius, the material properties of the liquid and the gas and the mean free path of gas molecules.

---

Monday, November 25, 2019 8:00AM - 10:36AM – Session H22 Drops: Heat Transfer and Evaporation I

8:00AM H22.00001 On the effect of wicking on droplet cooling\(^1\). MANUEL AULIANO, DAMIANO AULIANO, MARIA FERNANDINO, NTNU, PIETRO ASINARI, Politecnico di Torino, CARLOS DORAO, NTNU — Enhancing the droplet cooling is important for an efficient and safe design of thermal management applications, such as electronics, nuclear and aeronautics industry. For enhancing the droplet cooling heat transfer, it is desirable that the droplet spreads as much and as fast as possible. In this regard, tailoring the surface with micro/nanostructures is a promising approach that controls the surface wettability and the heat transfer performance. This work focuses on the effect of the wicking of super-hydrophilic stochastic Si nanowires on droplet evaporation from low to high temperatures. The research method consisted in fabricating Si nanowires with different heights and characterizing the wickability of the surfaces prior to the heat transfer experiments. The evaporation performance of the processed samples has been discussed in terms of the droplet evaporation time. A significant reduction of the evaporation time and shift of the Leidenfrost point were observed and attributed to the strong wicking provided by the nanostructured surfaces.

\(^1\)Research Council of Norway, NorFab, GE Healthcare
A Switchable Wettability Surface for Condensation Heat Transfer Manipulation

Jonathan Ludwicki, Paul Steen, Cornell University — The development of variable heat transfer devices is critical to enabling human exploration missions beyond low-Earth orbit (2015 NASA Technology Roadmaps). Here, we show the ability to manipulate condensation heat transfer performance via a surface with temperature-dependent wettability. Specifically, a nanostructured poly(N-isopropylacrylamide) surface was developed to switch the surface wettability between hydrophobicity and hydrophilicity via temperature stimulus. The hydrophobic surface condensation formation produces flooded dropwise condensation, which shows a two-times larger heat transfer coefficient versus filmwise condensation in the hydrophilic state. Such surfaces hold promise as a means to control heat transfer during flow condensation in spacecraft two-phase flow thermal control systems.

Supported by NASA Space Technology Research Fellowship 80NSSC17K0144.

Freezing a rivulet

Antoine Monier, Sorbonne Université, CNRS, UMR 7190, Institut Jean Le Rond d’Alembert; F-75005 Paris, France; Axl Huerre, Christophe Josserand, Laboratoire d’Hydrodynamique (LadHyX), UMR 7646 CNRS-Ecole Polytechnique, IP Paris, 91128 Palaiseau CEDEX, France; Thomas Don, Sorbonne Université, CNRS, UMR 7190, Institut Jean Le Rond d’Alembert; F-75005 Paris, France; Laboratoire d’Hydrodynamique (LadHyX), UMR 7646 CNRS-Ecole Polytechnique, IP Paris; 91128 Palaiseau, TEAM, Sorbonne Université, CNRS, UMR 7190, Institut Jean Le Rond d’Alembert; F-75005 Paris; France; Team — We study experimentally the solidification of a water rivulet flowing down an inclined plane cooled to subzero temperatures. The growth of the ice is described by two successive regimes with different dynamics. First, the ice grows as if the water was not flowing, well described by a classical 1D Stefan problem. Then, thermal convection from the constant water supply starts to play a role in the solidification process. Finally, the system reaches a stationary state where water flows on a peculiar ice structure. The surprising linear geometry of this structure, with distance from the water injection point, is explained through scaling arguments. The physical explanation relies on the growth of the water boundary layer with respect to the finite size of the water rivulet. When the thickness of the thermal boundary layer reaches the free surface, the surface temperature of the water decreases with the plane position, allowing us to recover the linear behavior of the ice structure.

8:39AM H22.00004 Ice patterns formation in freezing drop impacts

Thomas Seon, Institut d’Alembert, CNRS - Sorbonne Université, Virgile Thievenaz, Institut d’Alembert, Sorbonne Université, Christophe Josserand, LadHyX, CNRS - Ecole Polytechnique — We investigate the different shapes taken by a water drop frozen during its impact on a cold surface. The capillary hydrodynamics of a water film dewetting on ice, coupled with its vertical solidification, is quantitatively characterized. This allows us to understand and predict the formation of the emerging patterns. Finally, we show that this experiment enables a measurement of the contact angle of water on ice, whose value is still debated.

Frost pattern on surfaces with millimetric serrated features

Kyoo-Chul Park, Yuehan Yao, Tom Zhao, Emma Feldman, Christian Machado, Neelesh Pantankar, Northwestern University — Numerous studies have focused on a low surface energy coating and a micro/nanoscale surface texture to design functional surfaces that delay frost formation and reduce ice adhesion. However, the scientific challenges for long-term icophobic surfaces have not been fully addressed because of the complexity of mechanical freezing. Inspired by the suppressed frost formation on concave regions of natural leaves, here we report findings on the frosting process on surfaces with various serrated structures. Dropwise condensation, the first stage of frosting, is enhanced on the peaks and suppressed in the valleys when the wavy surface is exposed to humid air, causing frosting to initiate from the peak. The condensed droplets in the valley are then evaporated, resulting in a non-frost band. The effects of surface topography on the frost patterns are systematically studied by varying the serrated geometry defined as the vertex angle, and numerically modeling the spatial distribution of diffusion flux of water vapor on the wavy surface. Under different ambient humidity levels, the magnitudes of diffusion flux at the non-frost boundaries of the surfaces are nearly identical, implying that the critical value of diffusion flux is the key to understanding the non-frost pattern.

9:05AM H22.00006 Final fate of a Leidenfrost droplet: Explosion or takeoff

Sijia Lyu, Tsinghua University, Vargheese Mathai, Brown University, Yujie Wang, Tsinghua University, Benjamin Sobac, Pierre Colinet, University of Twente, Deletre Lohse, University of Twente, Chao Sun, Tsinghua University, Center for Combustion Energy, Department of Energy and Power Engineering, Team, Engineering Research Center Collaboration, Tips-Fluid Physics Collaboration, Physics of Fluids & Max Plank Center Collaboration — When a liquid droplet is placed on a very hot solid, it levitates on its own vapor layer, a phenomenon called the Leidenfrost effect. Although the mechanisms governing the droplet’s levitation have been explored, much is known about the fate of the Leidenfrost droplet. Here we report on the final stages of evaporation of Leidenfrost droplets. While initially small droplets tend to take off, unexpectedly, the initially large ones explode with a crack sound. We interpret these in the context of unavoidable droplet contaminants, which accumulate at the droplet-air interface, resulting in reduced evaporation rate, and contact with the substrate. We validate this hypothesis by introducing controlled amounts of microparticles and reveal a universal 1/3-scaling law for the dimensionless explosion radius versus contaminant fraction. Our findings open up new opportunities for controlling the duration and rate of Leidenfrost heat transfer and propulsion by tuning the droplet’s size and contamination.

Pattern formation from dried drops can be used to identify adulterated medicines

Yojana Carreon, Instituto de Investigaciones en Materiales, Universidad Nacional Autónoma de México; Ivan Cipriano-Urbano, Universidad Autónoma de Coahuila; Francisco Solorio-Ordaz, Jorge Gonzalez-Gutierrez, Facultad de Ingeniería, Universidad Nacional Autónoma de México, Roberto Zenit, Instituto de Investigaciones en Materiales, Universidad Nacional Autónoma de México — It is estimated that in Europe, from 2009 to 2012, 12.8 million of patients were exposed to counterfeit medicines. In Mexico, it is suspected that, between 2006 and 2013, many children suffering from cancer received distilled water instead of chemotherapeutic drugs. Fluid mechanics can be used to prevent such vile acts. Here we propose a simple method to detect the authenticity of medical substances, considering the patterns formed by dried droplets. The method is based on the structural analysis of patterns generated after the solvent of a drop of liquid medicine has evaporated over a solid substrate. We exploit the capillary flux and the ionic interactions as a natural aggregation mechanism to form patterns, which reveals unique structures for each active ingredient or drug. We demonstrate that the first-order statistics (FOS) and gray level co-occurrence matrix (GLCM) are sufficient metrics to distinguish among deposits generated by pure and adulterated drugs with an accuracy greater than 90%.

1Supported by NASA Space Technology Research Fellowship 80NSSC17K0144.

1Membership Pending’
9:31 AM H22.00008 Crystal Critters: Growth and ejection of crystals from heated, superhydrophobic surfaces, SAMANTHA MCBRIDE, HENRI LOUIS GIRARD, KRIPA VARANASI, Massachusetts Institute of Technology. Evaporating a drop of a volatile liquid containing a non-volatile solute leads to crystallization of said solute due to rising concentrations exceeding the solubility limit. When a drop of salty water is evaporated on a hydrophobic surface, “salt globes” that mirror the shape of the drop form due to nucleation at the air/water interface. Here, we present an unusual phenomenon in which salt globes grow from an evaporating drop on a heated superhydrophobic surface proceed to self-eject from that surface via growth of crystalline tubules. We show that this phenomenon is due to a specific superhydrophobic texture that combines minimal contact points and an assortment of channels that allow vapor to escape. A large temperature gradient across the height of the drop concentrates vaporization near the substrate, and escaping vapor at contact points between substrate and liquid leads to growth of crystalline tubules. These tubules grow into “legs” causing the entire salt globe — and any remaining water — to lift off from the surface. We call these structures composed of salt globes balanced on tubule legs “crystal critters” due to their resemblance to biological forms. Following complete evaporation, the crystal critters have minimal contact with the substrate and can be easily removed; and could find application for anti-fouling surfaces for spray heat exchangers.

1Supported by the NSF GRFP, MIT Martin Fellowship, and Equinor via the MIT Energy Initiative

9:44 AM H22.00009 Drop impact on a coated inclined surface, SUHAS RAO TAMVADA, VARUN KULKARNI, NIKHIL SUBHASH SHIRODA, GAURAV KULKARNI, SUSHANT ANAND, University of Illinois at Chicago. The impact of a drop onto a surface is useful in assessing the water repelling attributes of substrates with a given wettability. Most studies have focused on drop impact on horizontal flat surfaces however oblique impact of a droplet onto a surface, representing more accurate scenarios such as impact onto turbine blades leading to ice accretion and pesticide spraying has not been investigated. To this end we study the gentle deposition of a droplet at different temperatures on paraffin wax coated on an inclined copper surface. We measure the distance and average velocity of droplet until it comes to a halt to evaluate the role of the impact conditions and surface inclination on the observed sliding behavior. As we increase droplet temperature enhanced melting of the wax layer promotes sliding of the droplet to larger distances eventually being arrested by the viscous drag due to the underlying melted wax layer. Of significant interest is the deviation from the typical stick-slip motion reported on inclined hydrophobic surfaces. Our studies evaluate the efficacy of wax coatings in promoting self-cleaning on inclined surfaces and their possible use in applications which demand expedient shedding of droplets.

9:57 AM H22.00010 Stability of evaporating sessile drops comprising binary mixtures, ADAM WILLIAMS, School of Engineering, The University of Edinburgh, GEORGE KARAPETAS, Department of Chemical Engineering, Aristotle University of Thessaloniki, PEDRO TENG, Jan de Viron, University of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, The University of Edinburgh. Spreading and evaporation of a binary mixture sessile drop is a highly dynamic and complex process governed by the interplay between capillary stress, evaporation, hydrodynamic flow, mass diffusion and surface tension, with both thermal and solutal Marangoni stresses also present. We examine the behaviour and stability of volatile wetting ethanol-water drops deposited onto heated substrates using both experiments and modelling. We take a one-sided approach, utilising lubrication theory to obtain a base state before assessing stability by performing a linear stability analysis evoking the quasi-steady-state approximation. Evolution equations are derived for the interface height, temperature and concentration fields, assuming that the mixture comprises two ideally mixed volatile components with a surface tension linearly dependent on both temperature and concentration. Numerically, the contact line is avoided by releasing the drop over a precursor film. Results from both simulations and experiments indicate that concentration gradients in binary drops give rise to super-spread, and, at high ethanol concentrations, contact line instability. Our initial stability analysis also confirms that the process is highly unstable.

1EPSRC and EC-RISE-ThermaSMART

10:10 AM H22.00011 Dynamics of evaporating and non-evaporating drops falling in air, JORGE CHAVARRIA, UNAM, Mexico, SAUL PIEDRA, Conacyt Cidesi-Centa, Mexico, JUAN ARBENEGUI-TROYA, Oxford, UK, LILIA MARGARITA HERNANDEZ, ISTEM, Mexico, GUILLERMO HERNANDEZ-CRUZ, UNAM, Mexico, A ALFONSO CASTREJON-PITA, Oxford, UK, EDUARDO RAMOS, UNAM, Mexico. We report experimental observations of the dynamics of evaporating and non-evaporating drops falling in air under the influence of gravity. We use water and acetone drops of approximately 1 mm initial diameter. The experimental rig allows us to observe and acquire the motion for more than 2 m which corresponds to almost one second of flight-time and 80% of its terminal velocity. Under these conditions, evaporation reduces the initial volume by approximately 50% when acetone is used. We find that the motion is rectilinear up to the point where the Reynolds number is 190 and 220 for evaporating and non-evaporating drops respectively. We suggest that under this regime the motion becomes helicoidal in both cases. It is observed that this Reynolds number coincides with the formation of hairpin vortices in the wake. The geometrical properties of the wake of the evaporating drops are visualized using a Schlieren technique. We developed a simple model to predict the position as a function time of evaporating and non-evaporating falling drops in rectilinear motion regime.

1JC, SP, and AACP acknowledge the support of the Royal Society via a Newton Mobility Award No. NI17019. JC also acknowledges the support of CONACYT mobility grant.

10:23 AM H22.00012 Drying water droplets: Suppression of the coffee-stain effect by letting them dry on a thin oil film, YAXING LI, Physics of Fluids group, University of Twente, CHRISTIAN DIDDENS, TIM SECHERS, Physics of Fluids group, University of Twente; Eindhoven University of Technology, HERMAN WIJSHOFF, Oce Technologies, B. V.; Eindhoven University of Technology, MICHEL VERSLUIJS, Physics of Fluids group, University of Twente, DETLEF LOHSE, Physics of Fluids group, University of Twente; Max Planck Institute for Dynamics and Self-Organization. We systematically study the evaporation of a water microdroplet put on a thin oil film, both experimentally and theoretically. First, the absence of an intercalated film between droplets and substrates is demonstrated by interferometry. The interfacial energies between the droplet, the oil film and the solid surface are the key parameters to determine the wetting characteristics. During evaporation, we measure the flow field with pIV, which shows that it is controlled by the contact line behavior and the wetting state of the film with the droplet. Once the microdroplet contains particles, they accumulate during the evaporation process. We experimentally finds that the final deposition of the particles is determined by the flow and by the movement of the contact line. We derive an analytical expression for the radial velocity profile in the flow field near the substrate, which proves that the hindering of evaporation at the rim of the droplet by the non-volatile oil meniscus prevents the flow towards the edge, and therefore suppresses the “coffee-stain effect.” We finally demonstrate that the final particle deposition can be manipulated by tuning the surface energy of the droplet by adding a specific amount of a surfactant.
8:00AM H23.00001 Turbulent superstructures in a zero pressure gradient turbulent boundary layer for the Mach number range $0.3 - 3.0^1$. MATTHEW BROSS, SVEN SCHARNOWSKI, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — Meandering high- and low-momentum flow motions often called superstructures in turbulent boundary layers (TBLs) can extend up to several boundary layer thicknesses and contain a large portion of the layer’s turbulent kinetic energy. However, compared to the extensive number of incompressible investigations much less is known about the structural characteristics for compressible TBLs. Therefore, in this investigation TBLs on a flat plate over a range of Reynolds numbers and Mach numbers are considered in order to investigate the effect of compressibility on superstructures. Measurements are performed in the Trisonic Wind Tunnel Munich (TWM) for $0.3 < Ma < 3.0$ and a friction Reynolds number of $2700 < Re_f < 14600$ or $19800 < Re_{ij} = \rho u_i u_j \theta^2 / \mu_w < 40800$. Velocity fields are recorded using planar particle image velocimetry methods (PIV and stereo-PIV) in three perpendicular planes. Using multi-point statistical and spectrogram methods it was found that the streamwise variable lines associated with superstructures in the log-law layer slightly increase with Mach number and a distinct increase in the spanwise spacing of these structures was found for the supersonic cases when compared to the subsonic and transonic cases

1This work was supported by the Priority Programme SPP 1881 Turbulent Superstructures funded by the Deutsche Forschungsgemeinschaft project number KA1808/21-1.

8:13AM H23.00002 Optimal interpretation of measurements for enhanced-fidelity prediction of hypersonic boundary layer transition$^1$. DAVID BUCHTA, TAMER ZAKI, Johns Hopkins University — High-speed boundary-layer transition is extremely sensitive to the freestream disturbances which are often uncertain, thus compromising the accuracy of model predictions. To enhance the fidelity of simulations, we directly infuse them with measurements data. Our methodology is general and can be adopted with any simulation technique, e.g. parabolized stability equations or direct numerical simulations. An ensemble variational (EnVar) optimization is performed, whereby we determine the upstream flow that optimally reproduces the measurements. The cost function can account for our relative confidence in the model and the measurements, and judicious choice of the ensemble members improves convergence and reduces the prediction uncertainty. We demonstrate our data-augmented simulations for boundary-layer transition at Mach 4.5. Without prior knowledge of the freestream condition and using only wall measurements from an independent computation (true flow), all the relevant inflow disturbances are identified, and their amplitudes and phases are accurately predicted. We can then evaluate the entire flow field, beyond the original limited wall measurements. Our predicted flow compares favorably to the true unknown state, and discrepancies are analyzed in detail.

1This work was supported by the Air Force Office of Scientific Research (FA9550-19-1-0230)

8:26AM H23.00003 Velocity-Vorticity Correlation Structure (VVCS) in Transitional Compressible Turbulent Boundary Layer$^1$. JUN CHEN, SHI-YAO LI, ZHEN-SU SHE, State Key Lab. for Turb. & Complex Sys., FAZLE HUSSAIN, Dept. Mech. Engrg., Texas Tech. Univ. — Velocity-vorticity correlation structure (VVCS) is used to measure the geometry of vortices in the numerically simulated compressible boundary layer (BL) at $Ma=2.25$, 4.5, and 6.0. Wall normal vorticity represented by $VVCS_{ij}$ corresponds to the low-speed streaks, flanked by counter-rotating streamwise vortices identified by $VVCS_{ij}$. The ratio of spanwise size to spanwise spacing of $VVCS_{ij}$ decreases from 3 to unity during transition to turbulence, indicating low-speed streaks gradually populating in $x$. During transition, correlation coefficient, $R_{ij}$ near the wall decreases at first then increases up to 0.55, indicating that the well-arranged hairpins break up into streamwise vortices, as observed in visualization of this region. At each $y$, the length of spanwise vortices identified by $VVCS_{ij}$ decreases fast during transition at all $Ma$, indicating thickening of BL, while becoming nearly invariant in $x$ in the well-developed region, consistent with a self-similar turbulent BL predicted by our theory (She2018JFM). For both $VVCS_{ij}$ and $VVCS_{ij}$ the spanwise size of vortices transverse size in the developed flow region. These results affirm the predominant role of the multi-layered structure and suggest new possibilities for control of turbulent flow.

1Supported by National Nature Science Fund 11221062, 11425020 and by MOST 973 project 2009CB724100.

8:39AM H23.00004 The post-singularity structure in the boundary-layer flow induced by a rotating sphere. JIM DENIER, SOPHIE CALABRETTO, Macquarie University — The flow induced by a rotating sphere has long held the interest of fluid mechanists. This simple flow provides a paradigm for the study of colliding boundary layers; in the case of the sphere the boundary-layer collision occurs when fluid is advected from the poles to the equator. The collision process is the physical manifestation of a finite-time singularity in the boundary-layer equations. Here we will present new experimental and computational results which demonstrate that the collision process results in a smooth separation of flow within the boundary layer at the equator and into a radial jet. Our results demonstrate that existing theories for the post-singularity structure do not provide an accurate description of the flow.

8:52AM H23.00005 The Leading Edge theory: a new insight into the laminar Boundary Layer. MOHAMMAD GABR$^2$, Independent Researcher — The flow properties at the leading edge of a flat plate represent a singularity to the Blasius laminar boundary layer equations; by applying the diffusion equation where the velocity of a moving flat plate in a stationary fluid is diffused to the far field, the leading edge velocity profiles are studied. Experimental observations as well as the theoretical analysis show an exact Gaussian distribution curve as the original starting profile of the laminar flow. To conclude; the main key results are as follows:

1The velocity profiles at the leading edge of a flat plate are Gaussian Curves that grow in space and time; whereas the Blasius velocity profile is a part of the general Gaussian curves solution.

2A new method to calculate the friction drag is successfully tested, based on the displacement area of the leading edge velocity profile.

3In order to obtain the final physical proof of the new theory it is recommended to carry out experiments using an ultra-thin flat plate moving in a stationary fluid and measuring the velocity profiles at the leading edge with different measurement techniques.

1M.Sc.A. Mechanical Engineering, Polytechnique Montreal

9:05AM H23.00006 Boundary-layer flow of air over a falling soap film$^1$. YUNA HATTORI, RORY CERBUS, JULIO BARROS JR., PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology Graduate University — A falling soap film is a well-known experimental setup to realize two-dimensional flows in a laboratory. The soap film is invariably embedded in ambient air, which, in turn, is set to motion due to the falling film. We experimentally measure the velocity profile in the air using Particle Tracking Velocimetry (PTV). We find that the measured velocity profile conforms well to theoretical predictions using boundary-layer approximation. We discuss some implications of our results on the modeling of geophysical flows.

1Okinawa Institute of Science and Technology Graduate University
and biofilm like surfaces will be discussed and flow measurements will be presented.

velocimetry, and in situ measurement of the biofilm replacement with a line scanner. Comparisons of the resistance curves for the rigid replicas (i.e. non-evolving, such as faux fur), and their own three-dimensionally printed rigid replicas in a controlled environment. The surfaces will be exposed to flow raised additional questions. In this study, it is proposed to characterize biofilm like surfaces that have a more stable formation however, it is often attributed to the surface roughness, compliance, and the presence of undulating streamers in the flow boundary. Recent studies on living biofilms and rigid analogs have helped to address this very issue with acknowledgement that the biofilm evolution over time as it is exposed to flow raised additional questions. In this study, it is proposed to characterize biofilm like surfaces that have a more stable formation (i.e. non-evolving, such as faux fur), and their own three-dimensionally printed rigid replicas in a controlled environment. The surfaces will be evaluated in the Skin-Friction Flow Facility at the University of Michigan using streamwise pressure drop measurements, planar particle image velocimetry, and in situ measurement of the biofilm replacement with a line scanner. Comparisons of the resistance curves for the rigid replicas and biofilm like surfaces will be discussed and flow measurements will be presented.

9:31AM H23.00008 Augmenting Restricted Nonlinear Turbulence to Capture Scale Separation1, BENJAMIN MINNICK, DENNICE GAYME, Johns Hopkins University — The restricted nonlinear (RNL) dynamics comprises a streamwise constant mean flow interacting with a dynamically restricted perturbation field. This model aims to capture key features of wall-bounded turbulence with reduced computational expense. Constraining the nonlinear interactions leads to a reduced representation in which the RNL dynamics are supported by a small number of streamwise varying modes (non-zero streamwise Fourier coefficients) interacting with the streamwise mean, which captures statistical features of low Reynolds turbulence. To correctly capture momentum transfer at moderate Reynolds numbers where scale separation emerges, the streamwise varying modes must correspond to the small wave lengths associated with the outer-layer peak of the surrogate dissipation spectra. At higher Reynolds numbers where there is a clear separation of scales, we expect the model to require additional large scales consistent with triadic interactions of the small scales. Here, we simulate these interactions by including a large scale streamwise varying mode which is permitted to interact nonlinearly with small scale modes. We demonstrate that this augmented RNL (ARNL) model reproduces key flow statistics for Reynolds numbers as high as $Re_r \approx 5200$.

1This work is supported by ONR (N000141712649) and NSF (CBET 1652244).

9:44AM H23.00009 Convolutional neural networks for identifying coherent turbulent structures, ERIC JAGODINSKI, SIDDHARTHA VERMA, Florida Atlantic University — Identifying coherent structures and extreme events is key to understanding the physics of near-wall turbulent flows. Deep convolutional neural networks were originally developed for image recognition, but have immense potential for extracting features in multi-dimensional vector fields. Here, velocity fields extracted from Direct Numerical Simulations (DNS) of a periodic turbulent channel flow were input into a convolutional neural network in order to identify coherent structures and extreme events. The domain was reduced to a Minimal Flow Unit (MFU), as done in Jimenez (1991), and features were labeled for training the neural network using thresholding. Once trained, the network is able to correctly classify coherent structures and extreme events in the flow field.

9:57AM H23.00010 Dynamics and structure of backflow events in turbulent channels1, JOSE CARDESA-DUENAS, Univ Politecnica de Madrid, JASON MONTY, University of Melbourne, JULIO SORIA, Monash University, MIN CHONG, University of Melbourne — A statistical description of flow regions with negative streamwise velocity is provided based on simulations of turbulent plane channels in the range $547 \leq Re_r \leq 2003$. It is found that regions of backflow are attached and their density per surface area - in wall units - is an increasing function of $Re_r$. Their size distribution along the three coordinates reveals that, even though in the mean they appear to be circular in the wall-parallel plane, they tend to become more elongated in the spanwise direction after reaching a certain height. Time-tracking of backflow regions in a $Re_r = 934$ simulation showed they convect downstream at the mean velocity corresponding to $y^+ \approx 12$, they seldom interact with other backflow events; their statistical signature extends in the streamwise direction for at least 300 wall units and they result from a complex interaction between high and low spanwise vorticity regions far beyond the viscous sublayer. This could explain why some statistical aspects of these near-wall events do not scale in viscous units; they are dependent on the $Re_r$-dependent dynamics further away from the wall.

1This research was funded in part by the Multiflow project of the European Research Council.

10:10AM H23.00011 Turbulent inflow information generation for compressible boundary layers1, GUILLERMO ARAYA, HPCVLab, U. of Puerto Rico-Mayaguez, KENNETH JANSEN, U. of Colorado-Boulder — In this study, the dynamic rescaling-recycling approach for incompressible flows (J. Fluid Mech., 670, pp. 581-605, 2011) is extended to compressible spatially-developing boundary layers (SDTBL) for turbulent inflow conditions. Since the inlet boundary layer thickness is fixed, the inlet friction velocity is computed based on a power function of the momentum thickness, where the power exponent is calculated “on the fly” according to the flow solution downstream. Thus, there is no need of an empirical correlation as in other recycling methodologies. Additionally, the Morkovin’s Strong Reynolds Analogy (SRA) is used in the rescaling process of thermal fluctuations. The methodology is validated in a suite of Direct Numerical Simulation (DNS) of isothermal Zero-Pressure Gradient (ZPG) boundary layers at a Mach number of 2.86. The Reynolds number range is approximately 400-800, based on the freestream density, momentum thickness, freestream velocity and wall viscosity. Low/high order flow statistics are compared with wind tunnel experiments from the literature. Focus is given to the assessment of wall temperature on the thermal transport phenomena and the dynamics of thermal turbulent coherent structures.

1NSF-CAREER #1847241, AFOSR #FA9550-17-1-0051

Monday, November 25, 2019 8:00AM - 10:36AM
Session H24 Bubbles: Growth, Heat Transfer and Boiling I
606 - Prashant Valluri, University of Edinburgh
8:00 AM H24.00001 Viscous growth and rebound of confined bubbles. SEBASTIEN MICHELIN, LadHyX - Ecole Polytechnique, GIACOMO GALLINO, FRANCOIS GALLAIRE, LFMI - Ecole Polytechnique Fdrale de Lausanne, ERIC LAUGA, DAMTP - University of Cambridge — In many applications (e.g. bubble-powered micro-rockets), microscopic gas bubbles nucleate and grow in close proximity with a rigid bounding surface. This has profound hydrodynamic consequences, as the bubble is geometrically constrained and must translate away from the wall as its radius increases, in order to maintain mechanical equilibrium with the non uniform hydrodynamic stresses prevalent on the interface. This dynamic process is analyzed theoretically for a spherical inflating gas bubble in a viscous fluid, focusing specifically on the drainage dynamics of the thin lubricating film separating the bubble from the wall and for bubble surfaces of different physical nature, ranging from “clean” (i.e. stress-free) to “polluted” or rigid (i.e. slip-free) interfaces. Different bubble surface conditions lead to fundamentally different behaviors: bubbles may thus drain the lubrication film monotonically or bounce off the surface before eventually draining the film. A final universal regime (i.e. for all bubble surface conditions) is finally identified.

1 European Research Council through Grant Agreements 714027 (SM), 280117 (FG) and 682754 (EL)

8:13 AM H24.00002 Bubble Coalescence at the Free Surface. DANIEL SHAW, LUC DEIKE, Department of Mechanical and Aerospace Engineering, Princeton University — Bubble coalescence at a free surface occurs in our daily lives at the surface of drinks as well as at the surface of the ocean. While coalescence inside water has been largely investigated, the studies at the free surface remain scarce. In this talk, we study experimentally the coalescence of two bubbles at a free surface. Three primary regimes are identified. During an ‘attraction’ phase, the bubbles accelerate together due to the capillary distortion of the free-surface. As the bubbles draw near, the resistance provided by the fluid between them increases and slows the advancing bubbles. This ‘drainage’ regime begins when the relative velocity of the bubbles start decreasing and ends when the film is ruptured. ‘Confluence’, the third regime, is dominated by the rapid expansion of the neck separating the newly united air pockets. A balance of capillary, inertial, and viscous forces determine the dynamics of the newly-formed interface. Unlike coalescence in a bulk outer-fluid, the asymmetry created by the presence of the free surface alters previous models and presents new challenges for both measuring and modeling coalescence.

1 This work is supported by a grant from NSF Physical Oceanography to L.D. (Grant No. 1849762.)

8:26 AM H24.00003 Effect of Non-Uniform Circumferential Heat Fluxes and Orientation on Microchannel Flow Boiling – A Numerical Investigation. MARIUS VERMAAK, MOHAMMAD MOGHIMI, JOSUA MEYER, University of Pretoria, KHELLIL SEFIANE, PRASHANT VALLURI, University of Edinburgh — Flow boiling in microchannels exhibits incredibly high heat transfer characteristics, which could be revolutionary for multiple industries. However, the underlying physical phenomena that cause these characteristics are not well understood, especially the 3D effects of flow boiling in non-circular channels and the influence of confinement on bubbles. In this study, single bubble growth is considered in a high aspect ratio microchannel, hydraulic diameter 909 µm and aspect ratio 10, at varying gravitational orientations with only 1 face heated. While common microchannel theory neglects gravitational effects, at large aspect ratios gravity affects the hydrothermal characteristics of the flow. In particular, if microchannels are rotated along the axial direction. Bubble behavior after nucleation and its effect on heat transfer is investigated until slug formation. This investigation found that the heat transfer characteristics of the bottom heated case are the highest. In addition, the thermofluidic interaction between two sequential slugs moving down a microchannel is also presented. The results of this study have been validated against past numerical and current experimental work, some of which is being performed at the University of Edinburgh.

8:39 AM H24.00004 Nucleation of Plasmonic Bubbles in Binary Liquids. MARVIN DETERT, Physics of Fluids Group, University of Twente, The Netherlands, BINGLIN ZENG, YULIANG WANG, Robotics Institute, School of Mechanical Engineering and Automation, Beihang University, P.R. China, HAROLD J. W. ZANDVLIET, Physics of Interfaces and Nanomaterials, MESA+ Institute for Nanotechnology, University of Twente, The Netherlands, DETLEF LOHSE, Physics of Fluids Group, Max Planck Center Twente for Complex Fluid Dynamics, University of Twente, The Netherlands — When a noble metal, plasmonic nanoparticle is immersed in a liquid and irradiated with a laser of resonant frequency, it can heat rapidly and a vapor bubble can be nucleated. Bubbles generated this way are called plasmonic bubbles and have garnered a lot of attention due to their variety of applications. We want to disentangle the effect of various control parameters, such as the boiling temperature, the heat conductivity, the latent heat of vaporization etc., by measurements via ultra-highspeed imaging. A perfect candidate for these measurements are binary liquids, because their parameters can be tuned by their composition. We show both experimentally and theoretically that the time of bubble nucleation is determined by the total amount of dissolved gas. In contrast, the maximum volume of the bubble is governed by the energy needed for vaporization. Consequently, the bubble’s nucleation time and its maximal size can be tuned by varying the corresponding liquid parameters. We envisage that our findings will not only have important consequences for current applications, but might also result in new applications.

8:52 AM H24.00005 Nucleation and Growth of a Nanobubble on Rough Surfaces. SHANTANU MAHESHWARI, Shell Technology Centre Bangalore, COR VAN KRIJNISDIJK, Shell Technology Centre Amsterdam, SUCHISMITA SANYAL, Shell Technology Centre Bangalore, ALBERT HARVEY, Shell Technology Centre Houston. We study the nucleation and growth of a nanobubble on rough surfaces using molecular dynamics simulations. A nanobubble nucleates and grows by the virtue of a heterogeneous surface reaction which results in the production of gas molecules near the surface. We study the role of surface roughness in the nucleation and growth behaviour of a nanobubble. We perform simulations at various reaction rates and surface morphology, and quantified the growth dynamics of a nanobubble. Our simulations show that after the onset of nucleation, nanobubble grows rapidly with radius following t½ behaviour followed by diffusive growth regime which is marked by t¹/³ growth behaviour. This growth behaviour remains independent of surface roughness and reaction rates over the range considered in this study. We also quantified the oversaturation of gas required for the nucleation of a nanobubble and demonstrated its dependence on the surface morphology.

9:05 AM H24.00006 Near-wall bubble expansion and jetting collapse in generalized Newtonian fluids. JONATHAN FREUND, University of Illinois at Urbana-Champaign, RATNESH SHUKLA, Indian Institute of Science — The jetting dynamics of a gas bubble near a wall in a non-Newtonian fluid are investigated using axisymmetric simulations. The bubble gas is assumed homogeneous, with density and pressure related through a polytropic equation of state. An incompressible, Eulerian-frame, Navier-Stokes solver for generalized Newtonian fluids is used, with discretization modified to sharply represent the shear-free bubble-liquid interface. Simulations show both stabilization and destabilization due to non-Newtonian effects. In general, for fixed zero- and infinite-shear-rate viscosities, shear-thinning promotes and shear-thickening suppresses jet formation and impact. For a shear-thinning fluid, a threshold Carreau time scale A is found that suppresses both jetting and impact. Likewise, for shear-thickening, a minimum is found that suppresses both. The bubble-wall speed increases sharply with shear thinning and decreases for shear thickening. However, the bubble volume is far less sensitive, changing less than 50% for 0 < λ < ∞. The general trends, and particularly the high sensitivity of the jet speed to λ, suggest a criterion that could potentially protect tissue in biomedical applications and be used for high-strain-rate, large-deformation rheology.
Bubbles and droplets with mobile surfaces bounce stronger but coalesce faster. Surfaces can lead to coalescence owing to a second collision driven by the added mass. Head-on collision of two drops, where for certain parameter regime, we counter-intuitively see a full rebound for mobile surfaces, while immobile simulations with the Gerris software using extreme local grid refinement, which can reproduce the experiments. This allows us to simulate the immobile, whereas the final coalescence takes longer. Experiments with a buoyant droplet show similar results. The stronger rebound is due to lower viscous dissipation in the intervening thin film, during the collision when the surfaces are mobile. We also perform Volume-of-Fluid simulations with the Gerris software using extreme local grid refinement, which can reproduce the experiments. This allows us to simulate the head-on collision of two drops, where for certain parameter regime, we counter-intuitively see a full rebound for mobile surfaces, while immobile surfaces can lead to coalescence owing to a second collision driven by the added mass.

This work is funded by King Abdullah University of Science and Technology (KAUST). Supported by NASA, Grant: NNX16AQ77G. Computational resources by NASA high-end computing program.

A fundamental study of the channel shape impact on microchannel flow boiling via direct numerical simulations, Microchannel flow boiling is an attractive cooling solution for high-power density electronic devices. The coolant flows in a microevaporator, where many parallel channels are etched on a wafer substrate directly bonded on the surface to be refrigerated. Despite an extensive literature, there is still disagreement about the optimal shape of the channels cross-section which maximises heat transfer. Therefore, we have performed a fundamental study of the impact of the channel shape on the dynamics of elongated bubbles growing in microchannels and the associated heat transfer. We use a customised version of the Volume-Of-Fluid method in OpenFOAM, which includes a non-equilibrium evaporation model. Elongated bubbles are seeded at the upstream of a long microchannel heated with a constant heat flux. Channel aspect-ratios from 1 (square) to 8 (rectangular) are tested. We observe an essential impact of the perimetral distribution of the liquid film surrounding the bubble on the heat transfer patterns. Square channels exhibit higher heat transfer rates at low flow rates, while very thin liquid films are observed; flattened channels yield the best performance at larger flow rates, as they promote the formation of an extended liquid film covering a large fraction of the channel wall.

Optimising boiling by Surface Design from the Nanoscale Upwards. By constructing a heated surface at the molecular scale, we model the creation and growth of a bubble. In this talk, we present an introduction to MD, before showing nucleation results for a range of different surfaces. Nucleation rates are seen to depend on the details of the surface and we discuss insights provided by the MD approach. The challenge is to make these MD data relevant to larger scales, by linking these nanoscale bubbles to CFD simulations and optimising surface design to maximise nucleation. We discuss potential solutions to this problem, using multi-scale methods guided by a programme of experimental measurements.

Funding through RAEng/PETRONAS Research Chair for OKM

9:31AM H24.00008 Gravity Effects on Pool Boiling Heat Transfer. Effects of gravity on boiling heat transfer efficiency is of special interest due to its application in two-phase cooling systems for spacecraft and satellite components. Experimental investigations have identified trends in heat flux scaling that demonstrate two distinct boiling regimes dominated by buoyancy (BDB) in high gravity and surface tension (SDB) in low gravity. Regression models constructed from experimental data show that the transition between the two regimes is dependent on heater size and degree of subcooling. However, the bubble dynamics in this intermittent region are not very well understood. This serves as a motivation to identify heat flux mechanisms associated with bubble shape, size and merger using high-fidelity numerical techniques to increase scientific understanding of the process. In this talk, we will present results from our three-dimensional simulations, show a quantitative and qualitative agreement with experiments, and discuss bubble statistics that govern the scaling of heat flux in BDB, SDB and transition regimes.

Supported by NASA, Grant: NNX16AQ77G. Computational resources by NASA high-end computing program.
10:23AM H24.00012 Optimal control of the nonspherical oscillation of encapsulated microbubbles for biomedicine1, FATIH A. ARIFI, MICHAEL L. CALVISI, University of Colorado Colorado Springs — Encapsulated microbubbles (EMBs) consist of a gas core surrounded by a stabilizing shell comprised of lipid, polymer, or protein and are used for ultrasound imaging and medical therapies, such as drug and gene delivery. The nonspherical oscillation of EMBs is essential for their function as it can enhance the acoustic signature in medical imaging as well as the onset of rupture, which can affect drug/gene release. Therefore, the ability to control such oscillations can improve the efficacy of diagnosis and treatment mediated by EMBs, and reduce unwanted side effects. This talk discusses the use of optimal control theory to optimize the acoustic driving for a specific objective, such as maximizing the nonspherical subharmonic response to improve blood-tissue contrast, or exciting shape modes to incite bubble rupture or the formation of microjets, which can facilitate drug/gene uptake. These objectives are achieved through prescribing various cost functions that enhance the dynamic response of shape modes while minimizing overall acoustic energy input to improve patient safety. Single frequency, dual frequency, and broadband acoustic forcing schemes are explored and compared.

1This work was supported by the National Science Foundation CAREER Award 1653992.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H25 Minisymposium: Bubbles, Drops and Particles in Non-Newtonian Fluids 607 - Andrea Prosperetti, University of Houston

8:00AM H25.00001 Hydrodynamic interaction of a bubble pair ascending in-line in a viscoelastic liquid. ROBERTO ZENIT, Brown University — When the liquid in a bubbly flow is non Newtonian, a strong tendency to form large clusters and aggregates has been observed. In order to understand the mechanisms that lead to clustering in these liquids, we conduct experiments to determine the hydrodynamic interactions for a pair of bubbles rising in-line in a quiescent liquid. We fabricate viscoelastic fluids with shear-dependent viscosity with water-glycerin mixtures and polyacrylamide. Bubble pairs of different sizes are produced with capillaries of different inner diameters; the control of the bubble formation is achieved using a pulsatile syringe pump and a set of rapid-closing valves. The motion of bubbles is visualized with a high-speed camera. The results are contrasted with those from a Newtonian reference fluid. We observe that the interaction is significantly different from the so-called drafting-kissing-tumbling process that occurs for bubble pairs Newtonian liquids. In general, for viscoelastic fluids we observe no-kissing (how sad). Also, the bubbles do not drift apart indefinitely after interacting; instead, the bubble-pair continues to rise at a constant separation distance and angle. Furthermore, the interaction process appears to be mediated by the appearance, or not, of the negative wake behind the bubbles. Preliminary results will be shown and discussed.

8:26AM H25.00002 Particle-laden viscoelastic turbulence in channel flow1, TAMER ZAKI, AMIR ESTEGHAMATIAN, Johns Hopkins University — Individually, viscoelasticity and particles can appreciably modify channel-flow turbulence: The addition of polymers can quench the turbulent shear stress and enlarge the coherent streamwise velocity-perturbation structures; it can also cause the turbulence to cycle between an active and a hibernating state. In contrast, dilute particle concentrations can enhance the turbulent shear stress and weaken the streamwise velocity disturbances. When both viscoelasticity and the particle phase are present, the dynamics become much more intriguing as we demonstrate using direct numerical simulations of particle-laden viscoelastic turbulence in a channel. A dilute concentration (5%) of neutrally buoyant particles can aggressively amplify the cycles of hibernating and active viscoelastic turbulence. These cycles, in turn, modulate the particle migration between the channel centre and the near-wall region. Within the global dynamics is interesting micro-scale phenomenology: The particle-pair distribution function indicates that viscoelasticity leads to an exclusion corridor upstream and downstream of the particle, in the streamwise direction. An explanation is provided by contrasting the conditionally averaged fields around the particles in the Newtonian and viscoelastic conditions.

1This work was supported in part by the National Science Foundation.

8:52AM H25.00003 Exotic shapes and microscale structure of a bubble rising in hydrophobically modified alkali-soluble emulsion polymer solutions, MATSUHIRO OHTA, Tokushima University — The motion of single air bubbles rising through 1.3 and 2.1 wt% hydrophobically modified alkali-soluble emulsion polymer (HASE) solutions are experimentally examined. As reported in past research (Ohta et. al, 2015), a bubble rising in a HASE solution will have a shape that is distinct from a bubble rising in a typical non-Newtonian fluid. A bubble rising in a HASE solution can attain a shape with a very long thin trailing edge, long branched trailing edges, blade-shaped (two-dimensional thin plate shape) trailing edges, and more. By intuition, it is predicted that these distinct bubble shapes are formed due to the contribution of the elastic effect of HASE solutions. It has been discovered experimentally that the microstructure at the trailing edge of a rising bubble is intimately related to the concentration of HASE material in the liquid surrounding a bubble and the bubble size. The various HASE induced microstructures observed are categorized: (1) herring bone, (2) fish backbone, and (3) dendritically-expanded. It is hypothesized that the determination of one of these 3 shapes cannot be explained by straightforward surface tension considerations, and instead one must investigate localized interactions between the individual polymers in a HASE solution and the gas phase in the bubble.

9:18AM H25.00004 Nonlinear physics in bubble ‘buku-buku’ process with application to quasi-periodic volcanic eruptions, MIE ICHIHARA, Earthquake Research Institute, University of Tokyo, VALRIE VIDAL, Université de Lyon, Ecole Normale Supérieure — To mimic in the laboratory the acoustic phenomena due to bubbles rising in magma, we have investigated sounds produced by successive bubbles rising through fluid in a container. In a viscous Newtonian fluid, most of the bubbles were silent, while in the non-newtonian fluid all bubbles except the first one generated sound wave at their bursting on the fluid surface. We observed a modulation pattern of the acoustic waveform through time. Moreover, we found the existence of a precursor acoustic signal previous to each bursting. The time delay between this precursor and the bursting signal is well correlated with the bursting signal frequency content. Their joint modulation through time is driven by the memory of previous bubbles, especially the presence of small satellite bubbles trapped in the fluid due to the yield stress. At volcanoes, repetitive acoustic signals with modulation are often observed as seismic or atmospheric waves, which have been interpreted as a result of changes of the system parameters. Our experimental results have pointed out a new possibility that the non-Newtonian nature of magma with its memory effect spontaneously generates such modulations.
the solvent fluid can be neglected. A conservation law balancing pressure and elastic energy permits to calculate the thread thickness exactly.

and non-linear elasticity, we derive similarity equations for the full three-dimensional axisymmetric flow field in the limit that the viscosity of the thread profile, rescaled by the thread thickness, converges to a similarity solution. Using the correspondence between viscoelastic fluids and Oldroyd-B model, we find that...
8:05AM H26.00006 Shape optimization of stirrers for mixing binary fluids, MAXIMILIAN EGGL, PETER SCHMID, Imperial College London — Mixing is an omnipresent process in a wide-range of industrial applications. It thus seems prudent to devise techniques for optimizing mixing processes under time and energy constraints. To this end, we present a computational framework based on nonlinear direct-adjoint looping for the enhancement of mixing efficiency in a binary fluid system. The governing equations consist of the nonlinear Navier-Stokes equations, complemented by an evolution equation for a passive scalar. Immersed and moving stirrers are treated by a Brinkman-penalization technique, and the full system of equations is solved using a Fourier-based pseudospectral approach. The adjoint equations provide gradient and sensitivity information which is used to improve an initial mixing strategy, based on shape, rotational and path modifications. We utilise a Fourier-based approach for parameterising and optimising the embedded stirrers and consider a variety of geometries to achieve enhanced mixing efficiency. We study a restricted optimisation space by limiting the time for mixing and the rotational velocities of all stirrers. In all cases, non-intuitive shapes produced better mixing.

9:18AM H26.00007 A minimal model for riblet geometry optimization, MITUL LUHAR, ANDREW CHAVARIN, University of Southern California — Recent work shows that the forcing-response gain for resonant modes resembling the energetic near-wall cycle is a useful predictor of overall drag reduction performance over riblets. Riblet sizes and shapes identified as being optimal in previous high-fidelity simulations and experiments are found to minimize gain for this resonant mode. Building on this observation, we develop a geometry optimization framework for streamwise constant riblets. The shape and size of the riblets is parametrized using Bezier curves. A gradient-based search algorithm is employed to identify curve parameters that minimize amplification of the near-wall mode. Unsurprisingly, this procedure identifies scalloped and blade-like geometries as being optimal. Moreover, when scaled in inner units, the optimal sizes for these geometries are found to be insensitive to Reynolds number. Preliminary optimization results that take manufacturability constraints into account are presented briefly. Physical mechanisms responsible for performance deterioration are discussed in the context of the resonant-based predictions.

9:31AM H26.00008 Solution to shape design of unsteady natural convection fields to control time history of flow velocity, EIJJI KATAMINE, National Institute of Technology, Gifu College, TAKASHI AOKI, Nagaoka university of Technology — This paper presents a numerical solution to shape design of unsteady natural convection fields to control time history of flow velocity. The square error integral between the actual time history of flow velocity and the prescribed time history of flow velocity on the prescribed sub-domain during the specified period of time is used as the objective functional. Shape gradient of the shape design problem is derived theoretically using the Lagrange multiplier method, adjoint variable method, and the formulae of the material derivative. Reshaping is carried out by the projection method presented as an approach to solving shape optimization problems. Numerical analyses program for the shape design is developed based on FreeFem++, and the validity of proposed method is confirmed by results of 2D numerical analyses.

9:44AM H26.00009 A model-based investigation of the effect of actuator geometry on boundary layer flows, Iagal Gluzman, Dennise Gayme, Johns Hopkins University — An input-output approach is used to study actuated boundary layers arising from different types of input signals and actuator geometries. We obtain the manipulated flow fields through the Navier-Stokes equations linearized about a given base flow with the spatial geometry of the actuator represented as an array of localized input sources organized in the pattern of interest. The actuated fields are then obtained by superposing the response of each localized source in the array pattern, e.g., a saw tooth plasma actuator whose signal varies in intensity over the geometry. Actuation signals including a single pulse, a train of pulses, and a continuous input are modeled through analytical solutions of the LNS system. As an example, a steady-state step response is used to reproduce the stationary actuated flow-fields due to different plasma actuator configurations in transitional boundary layers. The model is found to reproduce the vortical and streamwise velocity structures obtained in experimental and simulation studies qualitatively well. Our results demonstrate the promise of such an analytical tool in determining beneficial actuator configurations prior to detailed experimental or high fidelity simulations, which are too costly for expensive parametric studies.

1 Supported by NSF (CBET 1652244) and ONR (N00014-18-1-2534)

9:57AM H26.00010 Adjoint-based Interfacial Control of Axisymmetric Viscous Drops, Alexandre Fikl, Daniel J. Bodony, Aerospace Department, University of Illinois at Urbana-Champaign — We develop a continuous adjoint formulation for the control of the deformation of a clean, neutrally buoyant droplet in Stokes flow. The focus is on constant surface tension-driven flows, where the interface is deformed with the local fluid velocity. We apply well-known results from the field of shape optimization to rigorously derive the optimality conditions for a wide range of interfacial problems. In the cases of interest, we make use of boundary integral methods as a natural choice for the numerical discretization of the flow variables. In the static case, our methodology is tested on several tracking-type cost functionals, corresponding to classic shape optimization problems. We show agreement with black-box finite difference-based gradients and accurate minimization of the cost functionals. Finally, we show that the methodology also applies to the control of the unsteady droplet deformation, controlled by external forcing in the form of the Capillary number.

1 This work was sponsored by the Office of Naval Research (ONR) as part of the Multidisciplinary University Research Initiatives (MURI) Program, under grant number N00014-16-1-2017.

10:10AM H26.00011 A new Drunken Sailor perspective and Spontaneously Singular Control approach for Lagrangian particles (buoyancy-controlled balloons) in highly stratified turbulence, Thomas Bewley, UCSD, Paolo Lucini, UNISA, Gianluca Meneghello, MIT — We will discuss recent progress in an ambitious project proposing a low-cost balloon observation system for sustained (in time), broadly distributed (in space), in-situ (between 1-km altitude), real-time (from data acquisition to NCAR within 20 minutes) measurement of hurricane development. The high density in space and time of measurements from such a robotic sensor vehicle swarm (100 or more vehicles persisting for 5 days) will be invaluable in improving our ability to estimate and forecast such extreme and dangerous atmospheric events. Recent work has focused on a decentralized strategy for rejecting small-scale disturbances of the sensor balloons (acting as Lagrangian tracers in the horizontal directions) that arise due to unresolved turbulent flowfield fluctuations with a concomitant $\omega^{-2}$ spectrum. This is done by modeling the balloon velocities (note: NOT positions) with a statistical random walk away from those planned by the coordinating MPC formulation. Implementing a realistic control penalty, $<|u|>$, in this setting gives a curious (and, practically implementable!) Spontaneously Singular Control strategy (on-off); this strategy is ultimately to be combined in a hierarchical setting with centralized MPC coordination of the large-scale balloon trajectories.
10:23AM H26.00012 Enhanced scalar transport through predictive reorientation of flow fields, RUUD LENSVELT, MICHEL SPEETJENS, HENK NIJMEIJER, University of Technology Eindhoven — Enhancements to scalar transport by fluid flow involves improved redistribution of heat/chemicals through advection. Improvements will be beneficial for a large span of industrial applications ranging from inline mixing in food production to subsurface resource extraction. A common feature in the applications of interest is reorientation of boundary driven laminar base flows to promote scalar transport. In conventional approaches a fixed, periodic reorientation scheme (in space or time) is optimized to accomplish chaotic advection. However, it is unclear whether such approaches produce the most effective propagation of transport in the presence of significant diffusion and/or chemical reactions. In this work, we explore an optimization scheme to predict the optimal orientation to enhance scalar transport over a certain time horizon. Spectral decomposition of the base flow allows for a compact model to ensure efficient prediction of the scalar field in this scheme. We investigate boundary heating of a cold fluid in a 2D circular domain with reorientation of the base flow based on both schemes. The optimization approach shows significant acceleration towards homogenization of the scalar field which demonstrates its potential to improve transport with reorientation of flow fields.

Monday, November 25, 2019 8:00AM - 10:36AM –
Session H27 Biological Fluid Dynamics: High Re Swimming I
609 - Leah Mendelson, Harvey Mudd College

8:00AM H27.00001 Flapping locomotion across a water-air interface1, LEAH MENDELSON, CHRISTOPHER PANIAGUA, ETHAN GREENBERG, WING-YEE LAW, Harvey Mudd College — Inspired by jumping fish, we investigate the propulsive performance of plates and hydrofoils that are simultaneously flapping and translating vertically out of the water. Compared to fully-submerged scenarios, additional considerations in this application include reduced force production during partially-submerged movements and interactions between the propulsion and the free surface. We explore trade-offs between thrust production, stability, and splash control when the actuator is partially-submerged. In particular, we consider whether a decaying sinusoidal motion profile is a viable strategy for producing useful amounts of thrust while mitigating lateral forces throughout the water-to-air transition. We also identify factors that determine the critical time and position during the translation out of the water where no further propulsion is beneficial.

1This work acknowledges support from the Norman F. Sprague III, M.D. Experiential Learning Fund

8:13AM H27.00002 Bubble PIV Measurements of Swimming Sea Lions2, GINO M. PERROTTA, THE George Washington University, FRANK E. FISH, West Chester University, MEGAN C. LEFTWICH, The George Washington University, DANIELLE S. ADAMS, West Chester University, JENIFER ZELIGS, STAFANI SKROVAN, SLEWTHS — California sea lions are among the most agile of swimming mammals. While most marine mammals swim with their hind appendages flippers or fluke, depending on the species sea lions use their foreflippers for propulsion and maneuvering. The sea lions’ propulsive stroke generates thrust by forming a jet between the flippers and the body and by dragging a starting vortex along the suction side of the flipper. Prior experiments using robotic flippers have shown these mechanisms to be possible, but no flow measurements around live sea lions previously existed with which to compare. In this work, the flow structures around swimming sea lions are observed using an adaptation of Particle Imaging Velocimetry. To accommodate the animals, it was necessary to use bubbles as seed particles and sunlight as illumination. Three trained adult California sea lions were guided to swim through an approximately-planar sheet of bubbles for a total of 173 repetitions. The captured videos were used to calculate bubble velocities, which were processed to isolate and inspect the flow velocities caused by the swimming sea lion. The methodology will be discussed and measured flow velocities will be presented.

2NSF CBET-1604876, ONR N00014-17-1-2448 and ONR N00014-17-1-2312

8:26AM H27.00003 Flow Structures Generated by a Robotic Sea Lion Foreflipper, ADITYA A. KULKARNI, ELI KASHI, GINO PERROTTA, MEGAN C. LEFTWICH, The George Washington University — Unlike most biological swimmers that rely on body/caudal fin (BCF) type of locomotion, a California sea lion produces thrust by moving its large foreflippers from above its head into a position abducted against its abdomen, a motion called a clap. This is followed by a long glide in a streamlined position. The flow structures resulting from this motion will not resemble the traditionally seen structures during BCF swimming, namely the reverse von Kármán street. Here, we use particle image velocimetry (PIV) to study the flow around an anatomically correct silicone flipper that is actuated by a servo motor. The flipper is mounted on a robotic platform and is programmed to go through the motions of a sea lion clap. The resulting data indicates that thrust is not produced through compression of fluid between the ventral side of the flipper and the body. Instead, the surrounding fluid is entrained by the upper surface of the flipper, producing vortices that run along the span and directly off the tip of the flipper. We also notice a cutoff frequency after which the efficiency of velocity production diminishes, which indicates the existence of an ideal ratio between rotational velocity and tip speed.

8:39AM H27.00004 A tendon-inspired adjustable-stiffness joint improves swimming speed and efficiency, QIANG ZHONG, JIANZHONG ZHU, HILARY BART-SMITH, DANIEL QUINN, University of Virginia — Fish dynamically control muscle stiffness to improve their swimming performance. The advantages of adjustable stiffness are only partially understood, because experiments on robotic tail fins have been limited to 3 Hz or less, whereas fish and fish-inspired robots can flap their tail fins at 10-15 Hz. We present here an actuator that flaps tail fins up to 7 Hz and uses a tunable spring to precisely control the stiffness of the peduncle (tail fin joint) in real-time. Our results show that dynamic stiffness control allows tuna-like fish to maintain high efficiency over a wide range of speeds (0 – 2.5BL/s). We tested this result on a multi-speed long-distance mission (500 m) and found that controlling stiffness while swimming can reduce energy consumption. Three-dimensional Particle Image Velocimetry illustrates what wake structures are responsible for improving efficiency, particularly when peduncle stiffness is optimized at high speeds. Understanding the flow physics governing adjustable tail stiffness at high swimming speeds could guide biological hypotheses about muscle control in fish and offer design ideas for fish-inspired underwater vehicles.
8:52AM H27.00005 What do fishes and fighter jets have in common?1, DANIEL QUINN, QIANG ZHONG, HAIBO DONG, University of Virginia, SMART FLUID SYSTEMS TEAM — Multi-fin systems, like fish or fish-inspired vehicles, are governed by unsteady three-dimensional interactions between their multiple fins. In particular, dorsal/anal fins have received much attention because they are just upstream of the main thrust-producing fin: the caudal (tail) fin. We used a tuna-inspired fish model with variable fin sharpness to study the interaction between elongated dorsal/anal fins and caudal fins. We found that the performance enhancement is stronger than previously thought (15% increase in swimming speed and 50% increase in swimming economy) and is governed by a three-dimensional Dorsal Fin-induced Crossflow that lowers the angle of attack on the caudal fin and promotes spanwise flow. Both simulations and multi-layer Particle Image Velocimetry reveal that the crossflow stabilizes the Leading Edge Vortex on the caudal fin, similar to how wing strakes prevent stall during fixed-wing aircraft maneuvers. Unlike other fin-fin interactions, this mechanism is phase-insensitive and offers a simple, passive solution for flow control over oscillating propulsors. Our results offer new insights into dorsal/anal fin shape and placement in fish and fish-inspired vehicles.

1 This work was supported by the Office of Naval Research under Program Director Dr. Robert Brizzolara, Award No. N00014-18-1-2537

9:05AM H27.00006 Aerodynamic Characteristics of Bio-Inspired Wings with Spanwise Waviness in a Turbulent Freestream, ALEXIA MARTINEZ-IBARRA, ROBERT FREEMAN, ISAAC CHOUTA-PALLI, The University of Texas - Rio Grande Valley — An experimental study is performed to investigate the effect of varying amplitude and wavelength of leading-edge tubercles on the aerodynamic and flow field characteristics of a NACA-0010 airfoil in flow with high freestream turbulence intensity of 4%. The study involved three airfoils – a smooth leading-edge, and two with a tubercle amplitude of 6% of chord, and with eight and four tubercles (varying wavelength) over the span. The aspect ratio of all airfoils was 2.0. The freestream velocities ranged from 16 to 40 m/s, with the corresponding chord-based Reynolds number varying from 160K to 412K. The angle of attack was varied from -6 to 30 degrees. The results show that the two tubercled airfoils achieved little to marginal performance enhancement pre-stall. In the post-stall regime, the lift coefficient of the longer wavelength tubercle airfoil was only marginally higher than the baseline while the smaller wavelength airfoil evinced the highest lift and delayed stall in the range of Reynolds numbers tested. The flow field data obtained using PIV showed large amount of turbulent mixing at leading edge of the tubercle airfoils, thus delaying flow separation, and leading to delayed stall as compared to the baseline airfoil.

9:18AM H27.00007 Optimal three-dimensional trajectory to generate maneuvering forces with a caudal fin of large aspect ratio, CECILIA HUERTAS-CERDEIRA, MORTEZA GHARIB, California Institute of Technology — The flapping motion of a caudal fin is an efficient method of generating thrust forces in an unmanned underwater vehicle. Simple pitching and heaving motions of this appendix, however, are not sufficient to achieve agile maneuvering of the vehicle. To address this deficiency, the use of a caudal fin that can perform large rotations around all three axes is explored. Due to the large number of possible trajectories attainable by such a mechanism, this study employs an experimental optimization procedure to obtain the most efficient three-dimensional trajectory that can generate a specified side-force value, equivalent to a turning moment. The optimal trajectory followed by a fin of large aspect ratio is presented and shown to be highly efficient. The trajectory is then experimentally analyzed in detail, and the use of fins of varying flexibility is considered, with increased flexibility being shown to be detrimental to the maneuvering performance of the fin.

9:31AM H27.00008 Swimming gait driven by proprioceptive feedback.1, JESUS SANCHEZ-RODRIGUEZ, CHRISTOPHE RAUFaste, MEDERIC ARGENTINA, UCA, CNRS, INPHYNI UMR 7010 — We have developed an elementary theoretical model of aquatic locomotion, based on [1] and [2]. We link the locomotion velocity to the kinematic of the foil. The amplitude and the beating frequency of the tail are still chosen by the swimmer and we would like to propose a simple mechanism which selects them. Here, we suppose that the tail motion proportionally depends on the normal force felt by its body. We have constructed a robotic compliant fish which is attached to a force sensor. We vary the feedback intensity and we measure the resulting thrust, amplitude and frequency. Our theoretical model accurately predicts the experimental outputs.


1 Funded by ANR-15-IDEX-01

9:44AM H27.00009 Burst-and-coast dynamics in zebrafish and tetrafish. 2, BENJAMIN THIRIA, RAMIRO GODOY-DIANA, ESPCI, FREDERIC LECHENault, LPENS, BILL FRANCOIS, ESPCI, GEN LI, DMITRY KOLOMENSKIY, Japan Agency for Marine-Earth Science and Technology, PMMH/LPENS TEAM, PMMH/LPENS TEAM — Swimming kinematics of small fish such as zebrafish or tetrafish are characterized by intermittent sequences consisting in an “active” swimming phase directly followed by a passive “coast” phase. This work is an attempt to characterize those sequences using several archetypal model experiments and models gathering hydrodynamics, statistics and behavioral sciences. We will focus on new results obtained from real fish experiments in free swimming and forced-gait configurations (using a controlled swimming channel). We will show how the statistics of these S2D sequences evolve with the conditions of the experiment as external flow conditions size of the habitat the maturity state of the fish (larva, juvenile or adult). We believe that these results will have direct implications on the design and implementation of biomimetic robotic systems.

9:57AM H27.00010 Hydrodynamic Interaction of Pitching Hydrofoils in Close Formatio, MICHAEL BOLTRI, OSCAR CURET, Florida Atlantic University — Swimming in close formation has the potential to improve swimming performance including swimming speed or power consumption. Numerical and physical models have shown that swimmers can take advantage of vortex shedding from other swimmers. However, most of the experimental work has been limited to few swimmers and tend to focus on the hydrodynamic interaction within the school. In this work, we developed an array of nine pitching hydrofoils (NACA0025) in close formation to examine the hydrodynamic interaction within the group and the wake generated by the group of foils. The pitching foils were tested in an inclined soap film to capture the flow structures generated by the foils. The foils were pitched at the quarter-chord with either sinusoidal or triangular wave patterns. The foils were tested with different frequency, amplitude, phase difference and spacing between them. We were able to capture the hydrodynamic interaction and the flow structure generated by the group. It appears that the flow structure generated by the group of foils is the result of a recombination of the vortex structures generated by the individual foils.

1This material is based upon work supported by the National Science Foundation under Grant No 1751548

2This material is based upon work supported by the National Science Foundation under Grant No 1751548
Reynolds number, using the point particle representation. In this study, we present turbulence statistics and the turbulence kinetic energy budget modulation expected in isothermal two-way coupled particle-laden flows. Using a multiphysics code to solve the coupled Navier-Stokes equations, to cause heated, and in turn transfer energy to a carrier fluid via convection. When the particle mass loading and radiative heating are large enough

JACOB WEST, SANJIVA LELE, Stanford University — In particle-based solar receivers, dense particles in a turbulent flow are radiatively

1

channel flow. which can improve undisturbed fluid velocity predictions in wall-bounded environments and apply this methodology to turbulent particle-laden

though a number of existing schemes are suitable on anisotropic grid arrangements which are typical of wall-bounded flows, we demonstrate

freely decay. In wall-bounded configurations, the disturbance must decay more rapidly—to satisfy no-slip and no-penetration at solid boundaries.

Having been developed to accurately estimate the undisturbed fluid velocity at particle locations for particle sizes up to and exceeding the grid

However, this estimate can be highly inaccurate when the particle size is not small compared with the mesh size. Promising new approaches

have been developed to accurately estimate the undisturbed fluid velocity at particle locations for particle sizes up to and exceeding the grid

spacing. One limitation of these methodologies is that they apply strictly in unbounded settings, where the computed disturbance flow can

freely decay. In wall-bounded configurations, the disturbance must decay more rapidly—to satisfy no-slip and no-penetration at solid boundaries.

Though a number of existing schemes are suitable on anisotropic grid arrangements which are typical of wall-bounded flows, we demonstrate

they nevertheless cannot accurately predict the undisturbed fluid velocity at all wall-normal separations. We present an alternative paradigm

which can improve undisturbed fluid velocity predictions in wall-bounded environments and apply this methodology to turbulent particle-laden

channel flow.

1Supported by the National Science Foundation (NSF) under grant No. CBET-1903312

10:23AM H27.00012 The role of edge curvature on the thrust force in a stingray inspired plan-form . RAVI CHAITHANYA MYSRA, PABLO VALDIVIA Y ALVARADO, Singapore University of Technology and Design — Optimizing the geometry of stingray inspired plan-forms for propulsive performance is of interest to design bio-inspired underwater vehicles and robots. Thrust generation is characterized by an exchange of momentum from plan-form to fluid as well as generation of edge vortices. Numerical simulations are performed at a Reynolds number of 500 on various geometries of the plan-form to better understand this phenomenon for a prescribed travelling wave. The plan-form surface area is kept constant while the geometry of the plan-form is varied from a rounded to a square shape by changing the curvature of the edge. As the curvature of the edge decreases, the thrust coefficient decreases. The effect of stingray inspired wave motion for various plan-form geometries with respect to thrust generation is analyzed in detail. The influence of edge curvature on the shape and strength of leading-edge vortices is characterized. The suction pressure and the generation of pressure due to exchange of momentum on the plan-form is also quantified.

Monday, November 25, 2019 8:00AM - 10:36AM –

Session H28 Particle laden Flows: Particle - turbulence Interactions 610 - Pedram Pak-seresht, Oregon State University

8:00AM H28.00001 Particle-laden Channel Flow with Strong Radiative Heating . JACOB WEST, SANJIVA LELE, Stanford University — In particle-based solar receivers, dense particles in a turbulent flow are radiatively heated, and in turn transfer energy to a carrier fluid via convection. When the particle mass loading and radiative heating are large enough to cause O(1) changes in carrier gas density and viscosity, this coupling introduces additional effects on the turbulence, beyond the turbulence modulation expected in isothermal two-way coupled particle-laden flows. Using a multiphysics code to solve the coupled Navier-Stokes equations, radiative transport equation, and Lagrangian particle trajectories, we perform direct numerical simulation of an irradiated channel flow at low Reynolds number, using the point particle representation. In this study, we present turbulence statistics and the turbulence kinetic energy budget for this flow at various particle mass loading and incident radiation levels. Funding support provided by US Department of Energy (DOE), Predictive Science Academic Alliance Program (PSAAP) II Center at Stanford: DE-NA-0002373

8:13AM H28.00002 Interaction between Saltating Particles and Turbulent Structures in Wall Region of Boundary Layer1 . WEI ZHU, Lanzhou University, DEPARTMENT OF MECHANICS, LANZHOU UNIVERSITY, KEY LABORATORY OF MECHANICS ON DISASTER AND ENVIRONMENTAL TEAM — Interaction between saltating particles and turbulent structures in the wall region of boundary layers was investigated through vertical 4-point measurements of wind fluctuations and sand particle counts synchronously at the Qingtu Lake Observation Array (QLOA) site. The measuring spots were positioned within the sand saltation layer at wall-normal heights of 0.15, 0.2, 0.3 and 0.5m respectively. Based on autocorrelation analysis on the spatial scale of coherent structures, the results show that saltating particles cause the streamwise length scale of coherent structures in the wall region decrease, and the effect introduced by moving particle-phase becomes more obvious as the wall-normal distance decreases. Spectrum analysis on the one-hour wind fluctuations with and without saltating particles shows that saltating particles significantly enhance small-scale turbulent fluctuations while the large-scale motions are weakened in the near wall region. These results indicate that saltating particles destroy the large-scale motions at the bottom of the logarithmic region of the atmospheric Surface Layer (ASL), which are dissipated into smaller-scale structures.

1This research was supported by the grants of the National Natural Science Foundation of China (nos. 11490553, 11702122).

8:26AM H28.00003 Comparison of Euler-Lagrange schemes in two-way coupled particle-laden channel flow1 . JEREMY HORWITZ, GIANNLUCA IACCARINO, JOHN EATON, ALI MANI, Stanford University — Considerable recent work to advance the state-of-the-art in Euler-Lagrange modelling of particle-laden flows has focused on the development of methods to estimate the undisturbed fluid velocity. This quantity is needed to calculate the drag force, which couples the motion of particles to the fluid. Historically, the undisturbed fluid velocity was estimated as the interpolated fluid velocity at the particle location. However, this estimate can be highly inaccurate when the particle size is not small compared with the mesh size. Promising new approaches have been developed to accurately estimate the undisturbed fluid velocity at particle locations for particle sizes up to and exceeding the grid spacing. One limitation of these methodologies is that they apply strictly in unbounded settings, where the computed disturbance flow can freely decay. In wall-bounded configurations, the disturbance must decay more rapidly—to satisfy no-slip and no-penetration at solid boundaries. Though a number of existing schemes are suitable on anisotropic grid arrangements which are typical of wall-bounded flows, we demonstrate they nevertheless cannot accurately predict the undisturbed fluid velocity at all wall-normal separations. We present an alternative paradigm which can improve undisturbed fluid velocity predictions in wall-bounded environments and apply this methodology to turbulent particle-laden channel flow.

1United States Department of Energy, PSAAP2
Transport and two-way coupling effect of inertial particles by large-scale and very-large-scale motions in turbulence

8:39AM H28.00004 Transport and two-way coupling effect of inertial particles by large-scale and very-large-scale motions in turbulence¹. GUOQUAN WANG, DAVID RICHTER, University of Notre Dame — Direct numerical simulations two-way coupled with inertial particles are used to investigate the particle distribution and two-way coupling effects associated with the large-scale motions (LSMs) and very-large-scale motions (VLSMs) in a channel flow at a moderate Reynolds number. One method of filtering the VLSMs from the flow is via artificial domain truncation, which alters the mean particle concentration profile and particle clustering due to the removal of VLSMs from a large domain simulation. In order to exclude possible correlation of the turbulence introduced by a small domain size with periodic boundary conditions, low- and high-pass filtering is performed during the simulation to isolate the particle interaction with different spatial scales. The particle interactions with different spatial scales are underpredicted without VLSMs, whereas the particle clustering and two-way coupling effects are mainly determined by particle coupling with LSMs. In the inner layer, the elongated streamwise anisotropic particle clustering can be reproduced by particles coupling solely with LSMs for low Stokes number particles. However, we do not observe similar particle clustering behavior in the outer layer as seen in the full simulation.

¹The authors acknowledge grants G00003613-ArmyW911NF17-0306 from the US Army Research Office and N00014-16-1-2472 from the Office of Naval Research.

Characterization of Inertial Particles in Turbulent Axisymmetric Wakes of a Porous Disk

8:52AM H28.00005 Characterization of Inertial Particles in Turbulent Axisymmetric Wakes of a Porous Disk. KRISTIN TRAVIS, SARAH E. SMITH, Portland State University, MICKAL BOURGOIN, ENS Lyon, HENDA DJERIDI, LEGI - Laboratoire des coulements Gophysiques et Industriels, RAL BAYON CAL, Portland State University, MARTIN OBLIGADO, LEGI - Laboratoire des coulements Gophysiques et Industriels. — Previous studies have suggested that dense particles such as dust and precipitation affect wind turbine performance. This study investigates the effects of micrometric inertial particles in the turbulent axisymmetric wake behind a stationary porous disk in a wind tunnel. Recent studies have explored turbulent wakes of stationary porous disks as analogs to the wakes of moving rotors yet inertial particles have not been considered. Phase Doppler interferometry and particle image velocimetry were implemented in the near wake of the turbulent wake to study the entrainment of inertial particles (mean diameter of 60μm) with a mean diameter of 60m). Reynolds numbers and water volume fractions were tested over ranges [Req (17.7 × 10^3 – 98.6 × 10^3)] and [Dv (3.9 × 10^-6 – 2.6 × 10^-5)] respectively. Results on the size distribution of particles within the wake, their settling velocity and the preferential concentration are discussed, showing a complex dynamics of such particles, as small particles are entrained and trapped in the near wake.

The role of discrete and carrier phase mechanical coupling in the inertial particles settling velocity

9:05AM H28.00006 The role of discrete and carrier phase mechanical coupling in the inertial particles settling velocity. DANIEL ODENS MORA, Universite Grenoble Alpes, CNRS, Grenoble-INP, LEGI, F-38000, Grenoble, France, and Department of Mechanical Engineering, University of Washington, MARTIN OBLIGADO. — Inertial particles settling velocity plays a non-negligible role in several environmental, industrial and biomedical applications, such as cloud formation and pollutant dispersion.

In this study, we have recorded via Laser Doppler Interferometry the vertical and horizontal velocities of sub-kolmogorov particles in a wind tunnel facility. We have explored a wide range of Taylor Reynolds numbers (between 40 and 650) by means of an active (actuated in two different modes), and a classical passive grid. We were able to estimate the turbulence modulation coming from the inertial particles presence with the help of a recent method proposed in the literature. The particle settling velocity data recorded for the different grids tested seems to better collapse when the turbulence modulation by the particles is included into the scalings.

¹Our work has been partially supported by the LabEx Tec21 (Investissements d’Avenir - Grant Agreement # ANR-11-LABX-0030), and by the ANR project ANR-15-IDEX-02.

Pair dispersions of particles during turbulence transition

9:18AM H28.00007 Pair dispersions of particles during turbulence transition. CHUNG-MIN LEE, California State University, Long Beach, ARMANN CYLFASON, Reykjavik University, FEDERICO TOSCHI, Eindhoven University of Technology — Small passive and inertial particles are present in a variety of natural or engineering flows, and their transportation and mixing by multi-scale turbulence is both of theoretical interest and practical importance. In many natural and industrial environments, however, the turbulent flow is in a transient state, as a prototype system, we investigate the transition from isotropic to anisotropic turbulence by looking at the influence of a transitioning turbulent flow on the statistical representation of the dynamics of the particles, and compare with results from homogeneous and isotropic flows. We conduct Direct Numerical Simulations of initially homogeneous turbulence, on which we suddenly impose a mean shear. The flows are initially seeded with passive and inertial particles, assumed of size that is sufficiently small and at sufficiently low seeding density so that their effect on the turbulent flow field can be neglected, and inter-particle dynamics can be ignored. Our interest is in the feedback forces exerted by particles and is damped further with increasing inertia of particles, that is, with increasing scales of particle motions.

In this study, we have recorded via Laser Doppler Interferometry the vertical and horizontal velocities of sub-kolmogorov particles in a wind tunnel facility. Recent studies have explored turbulent wakes of stationary porous disks as analogs to the wakes of moving rotors yet inertial particles have not been considered. Phase Doppler interferometry and particle image velocimetry were implemented in the near wake of the turbulent wake to study the entrainment of inertial particles (mean diameter of 60μm) with a mean diameter of 60m). Reynolds numbers and water volume fractions were tested over ranges [Req (17.7 × 10^3 – 98.6 × 10^3)] and [Dv (3.9 × 10^-6 – 2.6 × 10^-5)] respectively. Results on the size distribution of particles within the wake, their settling velocity and the preferential concentration are discussed, showing a complex dynamics of such particles, as small particles are entrained and trapped in the near wake.

A correction scheme for two-way coupled Euler-Lagrange wall-bounded flows

9:31AM H28.00008 A correction scheme for two-way coupled Euler-Lagrange wall-bounded flows. PEDRAM PAKSERESHT, Oregon State University, MAHDI ESMAILY, Cornell University, SOURABH V. APTE, Oregon State University — The accuracy of Euler-Lagrange point-particle methods in predicting the drag force reduces when the two phases are two-way coupled owing to the disturbance created by the point-particle force on the background fluid flow. Such disturbance produces an error since the drag closure models often rely on the slip velocity computed based on the undisturbed fluid velocity, which is not readily available. Recently, few approaches have been developed for unbounded flows only, where the disturbance field is not influenced by the no-slip boundary condition on walls. In this work, the correction scheme introduced by Esmaily & Horwitz (JCP, 2018) is extended to wall-bounded flows. It is shown that the newly developed model is generic for all types of flows with and without wall boundaries. The model is tested for a particle settling in a quiescent flow parallel to the wall at different wall distances. It is shown that ignoring the wall effects on correcting the disturbed velocity results in under prediction of settling velocity that is on the same order of magnitude of the lift force. We are able to estimate the turbulence modulation coming from the inertial particles presence with the help of a recent method proposed in the literature. The particle settling velocity data recorded for the different grids tested seems to better collapse when the turbulence modulation by the particles is included into the scalings.
Particle-turbulence and particle-wall interactions in a turbulent boundary layer

9:57AM H28.00010  YI HUI TEE, ELLEN LONGMIRE, University of Minnesota — The motions of finite-size spheres moving in a turbulent boundary layer are affected by both wall friction and coherent structures. 3D particle tracking experiments conducted at Re = 700 and 1300 with sphere diameters of 60 and 120 viscous units revealed two distinct sphere behaviors dependent on density ratio. Spheres held stationary on the wall with a density ratio of 1.03 always lifted off once released, translated without rotating, and subsequently fell back toward the wall. After the spheres had accelerated significantly, lift-off events with height larger than those following the initial lift-offs were observed frequently. These subsequent lift-off events are attributed to coherent structures in the flow. By contrast, spheres with a density ratio of 1.15 did not lift off initially, but slid along the wall once released. Further downstream, these spheres developed forward rotation as well as significant rotation about the wall-normal axis, and coincidently experienced repeated weak lift-offs. Wall friction was crucial in impeding the sphere acceleration as well as in helping initiate rotation. In the talk, PIV results on surrounding flow fields will be used to help explain the various particle-turbulence and particle-wall interactions.

Two-Phase Measurements Of Small Inertial Particles In High-Re Turbulent Boundary Layers

10:10AM H28.00011  TIM BERK, FILIPPO COLETTI, University of Minnesota — The interaction of small inertial particles with a turbulent boundary layer is of importance in a wealth of physical phenomena, e.g., transport of particles in the atmospheric boundary layer as well as industrial and biological applications. Our in-depth understanding is thwarted by the lack of comprehensive experiments. Here we perform two-phase measurements of microscopic inertial particles in high-Re turbulent boundary layers. Particles are injected in the boundary layer in the one-way coupled regime. Flow field measurements (PIV) and tracking of inertial particles (PTV) are performed simultaneously. This enables conditioned evaluation of fluid velocity at and around the inertial particles. The conditionally averaged flow around particles indicates the preferential sampling of specific flow structures by the inertial particles. This preferential sampling can help explain deviation of streamwise particle velocity from the mean fluid velocity and deviation of vertical particle velocity from the Stokes settling velocity. The fluid velocity at the particle location is an important term in, e.g., the advection-diffusion equation. Moreover, using the simultaneous two-phase measurements, widespread assumptions for estimating the concentration profile can be critically evaluated.

On the physical mechanism of the two-way interaction between polymers and near-wall turbulence

10:23AM H28.00012  JUNGHOON LEE, CHANGHOON LEE, Yonsei University — The two-way interaction between polymers and near-wall turbulence was investigated through direct numerical simulations (DNS) of turbulent channel flow coupled with Brownian dynamics simulations. The finitely extensible nonlinear elastic (FENE) dumbbell model was used to describe the motion of polymer molecules. The effect of polymers was included in the Navier-Stokes equations by exerting their reaction forces back on the fluid at the nearby eight grid points via a projection technique. In our simulations, the maximum extension of the FENE dumbbells does not exceed 0.6 in wall units, which is 100 times their equilibrium length. We examine how the polymer feedback forces affect the fluctuating motion of the fluid in coherent near-wall turbulent structures for various polymer relaxation times. Our results are generally consistent with previous DNS studies based on a continuum approach. We provide a more detailed picture of the physical mechanisms for turbulence modification by polymers through investigation on polymer stretch and orientation in the Lagrangian frame.

Towards Gamified Learning in Immersive Teaching of Fluid Mechanics

8:13AM H29.00002  NITESH BHATIA, Imperial College London, KAILYN BRYK, EVELYN SALAZAR, MIT, CHARLES RIGBY, OMAR MATAR, Imperial College London — FluidsVR is designed as a Virtual Reality (VR) platform for the exploration of transient and three-dimensional (3D) phenomena to promote deeper understanding of fluid mechanics. The platform serves as a learning medium by providing accurate and interactive models to the students based on the 3D data assets generated in-house by computational fluid mechanics researchers in the Matar Fluids Group. Students can visualize and interact with these assets using an intuitive graphical-user-interface. For improving the students’ affective domain, learning outcomes, and engagement levels, we are exploring the effectiveness of gamification in the context of teaching and learning. We have arranged the data assets in the increasing order of learning complexity and incorporated game mechanisms such as interactive questions, progress levels, scores and leader-boards to motivate students to further engagement and higher achievement. In this talk, we will cover this new gamification system for FluidsVR that was developed with the help of a group of undergraduate chemical engineering research students from Imperial and MIT.

3D+time flow visualization in virtual reality

8:00AM H29.00001  MELISSA GREEN, ZACKARY BOONE, Syracuse University — By viewing fluid dynamic isosurfaces in virtual reality (VR), issues associated with the rendering of three-dimensional objects on a two-dimensional screen can be addressed. In addition, viewing a variety of unsteady 3D data sets in VR opens up novel opportunities for education and community outreach. In this work, the vortex wake of a bio-inspired pitching panel is visualised using a three-dimensional structural model of Q-criterion isosurfaces rendered in virtual reality using the HTC Vive. Utilizing the Unity cross-platform gaming engine, a program can be developed to allow the user to “scroll” forward and backward in time to analyze the formation and shedding of vortices in the wake. The user can also toggle between different quantities, while keeping the time step constant, to analyze flow parameter relationships at specific times during flow development. In the current status of the game, all of the isosurfaces are rendered as objects to import, and all the relevant isosurface objects do need to be generated externally. There exists a great potential, however, to create a dynamic platform for an interactive and immersive research and education experience using off the shelf gaming systems. As part of this talk, we hope to provide a live demonstration with an HTC Vive.

Towards Gamified Learning in Immersive Teaching of Fluid Mechanics

8:13AM H29.00002  NITESH BHATIA, Imperial College London, KAILYN BRYK, EVELYN SALAZAR, MIT, CHARLES RIGBY, OMAR MATAR, Imperial College London — FluidsVR is designed as a Virtual Reality (VR) platform for the exploration of transient and three-dimensional (3D) phenomena to promote deeper understanding of fluid mechanics. The platform serves as a learning medium by providing accurate and interactive models to the students based on the 3D data assets generated in-house by computational fluid mechanics researchers in the Matar Fluids Group. Students can visualize and interact with these assets using an intuitive graphical-user-interface. For improving the students’ affective domain, learning outcomes, and engagement levels, we are exploring the effectiveness of gamification in the context of teaching and learning. We have arranged the data assets in the increasing order of learning complexity and incorporated game mechanisms such as interactive questions, progress levels, scores and leader-boards to motivate students to further engagement and higher achievement. In this talk, we will cover this new gamification system for FluidsVR that was developed with the help of a group of undergraduate chemical engineering research students from Imperial and MIT.

Supported by NSF (CBET-1510154).

Funding through Imperial College London Pedagogy Transformation programme is gratefully acknowledged.
8:26AM H29.00003 Use of Multi-sensory Immersive Technologies in Fluid Dynamics Education
LORENZO PICINALI, LYES KAHOUADJI, LACHLAN MASON, MARK SUTTON, NITESH BHATIA, ANDRIUS PATA-PAS, OMAR MATAR, Imperial College London — We present the recent virtual reality (VR) environment used in the Department of Chemical Engineering, Imperial College London, where both undergraduate and Masters-level students are able to "dive inside many classical examples of fluid mechanics (including Poiseuille flow, flow past a sphere [and associated vortex formation in the wake region], rising spherical-cap bubble, turbulent channel flow, and two-phase mixing in a stirred vessel) and explore the underlying physics. Three-dimensional CFD simulations are carried out to generate the flow field data for each flow, which are then imported into the VR. All physical fields from the CFD simulations, such as the magnitude of the velocity and stress components, as well as the pressure, are implemented in the VR environment via a sonification process where students are able to visualise and listen to chosen fields simultaneously. Our hypothesis is that this multi-sensory experience promotes a deeper understanding of the four-dimensional concepts underlying fluid dynamics.

1Funding through Imperial College London Pedagogy Transformation programme is gratefully acknowledged

8:39AM H29.00004 Conveying principles of fluid mechanics... through dance
JESSE CAPECELATRO, University of Michigan — Fluid mechanics is typically introduced to undergraduate engineering/physics students in their junior year, relying on knowledge from core courses in calculus and physics. Topics are traditionally introduced through tedious derivations that sometimes lack a clear conceptual interpretation. However, fluid mechanics is extremely visual, and visual solutions to classic fluid mechanics problems are highly aesthetic (for example, an oscillating wake past an obstacle, the swirl of a vortex, the chaotic motion of turbulence). In this talk, I will present an attempt at using dance to demonstrate fundamental principles of fluid mechanics. The objective is to create a visual demonstration of flow around a cylinder through a physics-constrained dance improvisation. This project involves a collaboration between Prof Capecelatro from the University of Michigan (UM) and dancers from the UM School of Music, Theatre & Dance. A series of dances were choreographed and filmed ranging from low Reynolds number (creeping) flow to high Reynolds number turbulence and inviscid (potential) flow. Details on the process and outcome of this collaborative effort will be presented, in addition to efforts to assess impact on student learning.

1This work was supported by the University Musical Society (UMS), University of Michigan Arts Engine, and University of Michigan School of Engineering

8:52AM H29.00005 Improving Scientific Visuals
NICOLE SHARP, Sharp Science Communications Consulting, LLC — There's truth to the old adage that a picture’s worth a thousand words. A well-designed visual can summarize your work, encapsulate an argument, or even draw media attention. Whether your research lends itself to exciting supplementary videos or your papers are made up mostly of graphical figures, it’s worth the effort to make your visuals as clean, understandable, and engaging as possible. This presentation focuses on what makes a good graphic and shares resources for improving your scientific visuals.

9:05AM H29.00006 Design and Development of Escape Room Based Interventions for Fluid Mechanics Courses
VRISHANK RAGHAV, MICHAEL MELNICK, Department of Aerospace Engineering, Auburn University — A new educational intervention tool based on the concept of escape rooms was designed and developed at Auburn University for use in the fundamentals of aerodynamics course. The details of the design for both conceptual and quantitative parts of the syllabi are discussed. The interventions consist of group-based and collaborative activities that could lead to a deeper level of engagement and understanding of fluid mechanics at the undergraduate level. The pilot implementation of these new tools and the re-design for different levels of undergraduate education and high-school outreach will be discussed.

9:18AM H29.00007 Using Coffee to Teach Fluid Dynamics
WILLIAM RISTENPART, University of California Davis — Fluid dynamics courses at the undergraduate level typically require a full year of calculus and physics as prerequisites. Accordingly, it is typically considered difficult to teach either first-year students or non-STEM (general education) students much about fluid dynamics. In this talk, we describe how brewing coffee serves as an engaging and hands-on introduction to fluid dynamics for students who have no calculus or physics background. Specifically, we use an AeroPress coffee brewer to introduce Darcy’s law and the key concepts of pressure drop, flow rate, viscosity, and permeability. Students apply a pressure by hand to a brewer placed on a standard bathroom scale, while simultaneously measuring the average flow rate out of the brewer. Measuring the thickness \( L \) of the spent moist grounds allows students to plot the flow rate versus the applied pressure drop \( P/L \) for different applied pressures and/or coffee masses, typically yielding a nice linear trend. Experiments with different grind sizes allows the impact of permeability to be explored quantitatively by linear regression. Importantly, the students can also taste the impact of the flow rate on the sensory qualities of the resulting brew, providing a memorable and highly caffeinated learning experience.

9:31AM H29.00008 Developing a New CFD Course Based on Open Source Tools: Design Experience and Student Outcomes
DEBANJAN MUKHERJEE, University of Colorado Boulder — Computational fluid dynamics (CFD) education comprises a fundamental component in training modern engineering students. CFD educators commonly explore fundamental aspects of the underlying methodology in conjunction with hands-on experience in using CFD techniques via commercial packages and tools. In addition to the (often significant) associated costs and hardware/system requirements associated with these tools, striking the right level of exposure to the intricacies behind these packages becomes critical to avoid student perception of these tools as a black box. Harnessing the open source wave provides a suitable alternative, enabling greater exposure to the underlying methods while providing the valuable hands-on simulation experience. However, integration of open source tools into educational components requires careful design considerations. We recently developed a new CFD technical elective in the mechanical engineering curriculum at the University of Colorado Boulder. Our course was successfully developed entirely based on open-source software tools and components. The class was composed of graduate and senior undergraduate students. Here we will share our experience designing the course, examples of student projects, and student learning outcomes.
Projects in Fluids Courses Made Easy (for You)  
JEAN HERTZBERG, University of Colorado Boulder — Projects can provide a number of significant benefits to students in fluids courses at both the graduate and undergraduate levels, but they can be difficult to implement, particularly in large class sections. A major problem is assessment of written project reports, which is very time consuming. Solutions include assigning projects to small teams, formed automatically by CATME.org based on student schedules. This also alleviates the #1 student objection to team projects: scheduling. Other demographics can be used in team formation to ensure inclusiveness. Students are also fearful of the unknown time requirements of a project. A scaffolding approach, breaking the project into small pieces, due at intervals through the semester helps students gain confidence and avoid procrastination. A detailed rubric is essential; it clearly states your expectations and provides a guide to assessment, allowing peer evaluation, the other essential ingredient that saves your time. Peer evaluation must be supported by class time spent on teaching constructive criticism. Anonymous peer evaluation is supported in learning management systems such as Canvas. Automatic plagiarism detection (Turnitin.com) prevents re-use of previous reports, and can be used to teach proper citation behavior. Allowing students to select their own project topics (from a list at least) promotes improved attitudes towards fluid mechanics, including the relevance of fluid physics in their daily lives.

Teardowns: students’ ability to identify components in actual systems  
MARIA-ISABEL CARNASCIALI, University of New Haven — The typical sequence of thermo-fluids related courses within mechanical engineering programs involves a thermodynamics, a fluid dynamics, and a heat transfer course. In most instances, these are accompanied by a lab component or supplemented with a separate lab. The intent of these hands-on courses (or supplements) is to broaden exposure to physical components and systems among several other learning outcomes. Students’ ability to make connections between the concepts presented in the various thermo-fluid courses and the physical parts/components with which they interact are essential to their long-term ability to function as engineers. However, these physical examples of system implementation represent but a limited view into the large variations that are currently present in and use in industry. Providing small, manageable components for students to investigate students’ ability to identify individual components but also intended purpose of the component within the system. Participants were asked to draft concept maps and system diagrams for a selected item; then proceeded to teardown the item. Participants included those who had and had not completed the lab component. Results provide insight to gaps in students knowledge and reinforce the need to show real systems as opposed to relying on system schematics or sketches.

The secret life of fluid dynamics in the patent world  
ISMAIL HAMEDUDDIN, Norton-Rose-Fulbright — A significant proportion of technical knowledge is only to be found in patent publications (80 percent of all technical knowledge, by some accounts). Fluid dynamics, often viewed as an abstruse discipline, has a significant and visible life in the patent world, with an entire CPC subclass devoted to it. On the other hand, scientific journals are replete with examples of well-meaning academics giving away perfectly patentable ideas, often to be trivially commercialized by others for unseemly gains. The patent literature is complementary, rather than an alternative, to peer-reviewed publications and is a tremendous resource for 21st century academics; it can be instrumental for sparking creativity, generating ideas for proposals, and for an insider look into the industrial/commercial applications and relevance of fluid dynamics. It may also be useful in battling fatalistic resignation among aspiring academics, which is often inadvertently induced by the well-meaning Clay Math Institute. The expense of obtaining (or even filing) a patent generally ensures the expression of commercially useful ideas in patent publications, even if they may not be expressed in the most articulate and persuasive manner.

Science Communication and Community: Entry Points to the Physical Sciences  
FRANCESCA BERNARDI, Florida State University, KATRINA MORGAN, Northwestern University — Students from underrepresented groups face numerous barriers of entry in STEM, especially in the hard sciences. Fostering a community of young people who are interested in science, but do not necessarily see themselves as possible scientists, can help them feel welcome in these fields and help build their confidence. Outreach events aligning science communication with scientific disciplines can be a point of entry for such students, who often find events with a communication aspect to be less intimidating than those focused exclusively on science activities. Incorporating communication aspects in science outreach serves several purposes: It encourages the participation of non-traditional STEM students by building a community of young people passionate about science and communicating it to the public; it helps educate these students to become better scientists, teaching them communication skills as well as challenging STEM topics; and it shows them the importance of focusing on both aspects. Experiences will be reported from Girls Talk Math, a free day camp for female and gender non-conforming high-schoolers focused on Mathematics, Physics, and media.
8:52AM H30.00003 From Depicting to Deploying Fluids in Art. ANDRZEJ HERCZYNSKI, Boston College — Water appears in ancient Greek and Roman art, on vases, in frescoes, and in mosaics of swimming fish or boats moving across the sea by oar or sail. Beginning in the Middle Ages, images of water in motion, waves, waterfalls, and wine or milk pouring from vessels, appear in landscapes and domestic scenes. Italian and Flemish Renaissance masters endeavored to render nature with the utmost attention to detail and were able, when their subject called for it, to capture fluid dynamics with astonishing precision. Including fluid effects has also allowed artists to convey motion in the static medium of painting or sculpture. Nevertheless, convincing representation of liquid flows, especially oscillatory or turbulent, has remained a challenge. The invention of non-figurative art proved liberating, leading Abstract Expressionist painters to adopt fluid phenomena themselves – jets, drips, sprays, and instabilities of liquid pigment – in the creative process. This talk offers a brief review of the quest to naturalistically depict fluids, and the alternative tactic of modern artists who learned to deploy fluids, endowing their abstractions with nature’s own patterns.

9:18AM H30.00004 Pattern formation in watercolor paintings. JORGE GONZALEZ-GUTIERREZ, Facultad de Ingeniería-Universidad Nacional Autónoma de México — The restoration of a masterpiece is essential to preserve its historical and commercial value. With the aim to find physical laws that allow us to predict the complex phenomena that emerge during the materialization of artistic work, we explore on the effect of the pigment concentration and paper humidity on the pattern formation driven by evaporation of droplets of watercolor paintings. These control parameters induce the formation of color gradients, stratifications, flat regions, borders, dendritic shapes, and radial tips. Interestingly, the droplet evaporation on dry paper form coffee rings stains regardless of the nature of the pigment. The mean pixel intensity of such stains follows an exponential function that saturates at high concentration, while the thickness of the coffee ring increase for watercolor inks containing colloidal particles and does not change for non-colloidal. The experiments reveal that water distribution on the paper surface, and not on the density of water on the paper, determine the structural characteristics of watercolor stains. We show evidence that the cornerstone in the creation of complex patterns in watercolor paintings is driven by the coffee ring effect and imbibition processes. Our findings could help with the correct application of restoration processes to preserve the heritage value of watercolor artwork.

9:31AM H31.00001 Geometry and Hydrodynamics of Flagellar Bundles1. MÁRIA TATULEA-CODREAN, ERIC LAUGA, DAMTP, University of Cambridge — Most motile bacteria exploit chirality in order to break the symmetry of low Reynolds number flows and generate propulsion. Spirochaetes have developed corkscrew-shaped bodies, while bacteria with simpler bodies can assemble and actuate helical flagellar filaments. In the case of multi-flagellated species, the bacterium can bundle and unbundle its flagellar filaments in order to swim in a straight line or change direction, respectively. This process is central to their run-and-tumble mobility. The hydrodynamic benefit of having multiple filaments, however, is associated with an increasing risk of tangling within the bundle. At one extreme, we know that straight flagellar filaments could not intertwine, but neither could they propel the cell forward in a viscous fluid. Similarly, one filament could never tangle on its own, but neither could it generate a propulsive force as large as a bundle of flagellar filaments. In this talk, we will present some recent theoretical results about the role that flagellar geometry and number play in the robustness of forming a tangle-free bundle and the hydrodynamic efficiency thereof.

1We gratefully acknowledge funding from the George and Lillian Schiff Foundation (award to MTC) and the European Research Council under the European Union’s Horizon 2020 research and innovation programme (grant agreement 682754 to EL).

Monday, November 25, 2019 8:00AM - 10:36AM – Session H31 Biological Fluid Dynamics: Locomotion Flagella 613 - Henry Fu, University of Utah

8:00AM H31.00001 Geometry and Hydrodynamics of Flagellar Bundles1. MÁRIA TATULEA-CODREAN, ERIC LAUGA, DAMTP, University of Cambridge — Most motile bacteria exploit chirality in order to break the symmetry of low Reynolds number flows and generate propulsion. Spirochaetes have developed corkscrew-shaped bodies, while bacteria with simpler bodies can assemble and actuate helical flagellar filaments. In the case of multi-flagellated species, the bacterium can bundle and unbundle its flagellar filaments in order to swim in a straight line or change direction, respectively. This process is central to their run-and-tumble mobility. The hydrodynamic benefit of having multiple filaments, however, is associated with an increasing risk of tangling within the bundle. At one extreme, we know that straight flagellar filaments could not intertwine, but neither could they propel the cell forward in a viscous fluid. Similarly, one filament could never tangle on its own, but neither could it generate a propulsive force as large as a bundle of flagellar filaments. In this talk, we will present some recent theoretical results about the role that flagellar geometry and number play in the robustness of forming a tangle-free bundle and the hydrodynamic efficiency thereof.

1We gratefully acknowledge funding from the George and Lillian Schiff Foundation (award to MTC) and the European Research Council under the European Union’s Horizon 2020 research and innovation programme (grant agreement 682754 to EL).

8:13AM H31.00002 Synchronization modes of active microfilaments. YI MAN, EVA KANSO, University of Southern California — Biological filaments are rarely found in isolation. Eukaryotic flagella beat in synchrony, an array of cilia generate the phase-locking metachronal wave. Experimental and computational studies provide a body of evidence that active filaments can synchronize through hydrodynamic coupling only. Previous theoretical models mostly address interactions in a far-field regime, where the interfilamentous distance h is much larger than the filament length L. Here, we employ a simple active filament model and combine it with an asymptotic result of hydrodynamic interactions in a more biologically-relevant regime where h/L ≪ 1. By varying the activity and coupling strength, we find three synchronization modes: in-phase, anti-phase, and a new mode which we refer to as asymmetric synchronization. We map the basins of attraction of these modes and find bistable in-phase and anti-phase synchronization when the coupling is strong. Furthermore, we present a thorough Floquet-type stability analysis to show the evolution of the phase difference between two filaments. The Floquet analysis proves the existence of the new mode of asymmetric synchronization, and it reveals a time scale it takes to reach the synchronized equilibria.
8:26AM H31.00003 How do colonial micro-algae swim towards light? , HELENE DE MALEPRADE, University of Cambridge, FREDERIC MOISY, Paris-Sud University, TAKUJI ISHIKAWA, Tohoku University, RAYMOND E. GOLDSTEIN, University of Cambridge — Microscopic algae are commonly found in mud, puddles or lakes, and show great diversity in structural complexity. One of the simplest algae encountered is the unicellular ‘Chlamydomonas’, exhibiting two flagella whose beating enables them to swim in a breast stroke. One also finds ‘Gonium pectorale’, a colony made of 16 Chlamydomonas-like cells arranged in two concentric squares, with all flagella on one side of the plate. These colonies are among the first multicellular algae and their study offers an insight into the evolution from unicellular to coherent multicellular behaviour. Algae, like plants, get energy from photosynthesis: Gonium colonies take advantage of their motility to swim towards light, efficiently reorienting within a couple of seconds. However, the mechanism of this phototactic behaviour is not yet understood: how do all 16 cells individually produce a coherent collective response? How are the flagella modulated to create an asymmetry in the swimming pattern, and how does that lead to reorientation? We experimentally investigate the phototaxis of Gonium, analysing their reorientation trajectory towards light. We compare those results to an analytical model and numerical simulations, describing with high precision the reorientation process.

8:39AM H31.00004 Bending Stiffness and Critical Forces for Polymorphic Transformations of Salmonella Flagella Measured in a Microfluidic Channel , HOSSEIN MOGHIMIFAR, Department of Mechanical Engineering, University of Utah — Bacterial flagella are internally-driven flexible filaments that display complex beating patterns. We develop a method to reconstruct the 3D geometry of a bacterial flagellum from fluorescence microscopy images with sub-pixel accuracy. The flagellar geometry is specified as a helix with pitch, radius, and axis direction that vary along its length. The expected image of the geometry is generated using point spread function. For each flow rate, we find the best-fit flagellar geometry by minimizing the pixel-to-pixel intensity difference between the generated and the microscopic image using a genetic algorithm. From the geometry of the flagellum and the known flow in the microchannel, we determined the forces on the flagellum using a boundary element method and found the critical force that caused the transformation. We also use a Kirchoff rod model to find the bending stiffness of the flagellum.

8:52AM H31.00005 Time-dependent hook flexibilities in run-reverse-flick motility, MEHDI JABBARZADEH, HENRY C. FU, University of Utah — The deformation of the hook and flagellum affects bacterial motility in run-reverse-flick motility of single-flagellated bacteria. Previously, we have modeled the initiation of a flick, in which the flagellum makes a large off-axis motion, by an efficient linear spring model with a rigid cell body and flagellum while neglecting hydrodynamic interactions between the cell body and flagellum. However, a complete flick event involves bending of both the hook and flagellum as well as a time-varying hook stiffness. Here, we study the dynamic bending of the hook and flagellar filament during run-reverse and flick motility of single flagellated bacteria. We develop an accurate and more efficient numerical approach to model the dynamics of free-swimming bacteria that includes flexibility of both the hook and flagellum. Using numerical models, we are able to constrain the time-dependent flexibility of the hook during run-reverse-flick motility. We compare results from rigid body simulations to the flexible flagellar filaments. Finally, we simulate complete flick events, investigating the buckling angle and reorientations of the swimming cells due to time-dependent hook stiffness.

9:05AM H31.00006 Hook flexibility and body trajectories of swimming microorganisms, ZONGHAO ZOU, WILSON LOUGH, SAVERIO SPAGNOLIE, University of Wisconsin-Madison — The flexibility of the hook connecting the bacterial flagellum to the cell body, and an associated buckling instability, is believed to play an important role in microorganism locomotion. We consider a simplified model for the flagellum-cell dynamics and solve analytically for the flagellum orientation and cell trajectory through space. This hook flexibility affects the swimming path of the cell body and flagella. We solve the problem for different geometries and cell velocities. We show that the model can reproduce key features of the swimming paths for a wide range of parameters. We also consider the effect of cell stiffness on the swimming paths and show that the cell stiffness can significantly affect the swimming dynamics. Our results provide insights into the role of hook flexibility in microorganism locomotion and suggest new experimental methods to study the role of hook flexibility in swimming.

9:18AM H31.00007 ABSTRACT WITHDRAWN —

9:31AM H31.00008 Estimation of Internal Power Distribution in Sperm Flagella from Measurements of Beat Patterns , ASHWIN NANDAGIRI, IIITB-Monash Research Academy, India, AVINASH GAIKWAD, School of Biological Sciences, Monash University, Australia, DAVID POTTER, Monash Microscopy Institute, Monash University, Australia, JULIO SORIA, Department of Mechanical & Aerospace Engineering, Monash University, Australia, MOIRA OBRYAN, School of Biological Sciences, Monash University, Australia, SAMEER JADHAV, Department of Chemical Engineering, Indian Institute of Technology Bombay, India, RANGANATHAN PRABHAKAR, Department of Mechanical & Aerospace Engineering, Monash University, Australia — Sperm flagella are internally-driven flexible filaments that display complex beating patterns. We estimate energetics of the internal driving from measurements of beat patterns. A large number of beat cycles (40) of mouse sperm tethered at their heads are recorded using high-speed, high-resolution microscopy. Flagellar centrelines are digitally extracted using image processing techniques. Proper Orthogonal Decomposition (POD) is used to represent the beat cycle data in a compact form and obtain an average representative beat cycle. The Kirchhoff theory for inextensible, elastic, rods is adapted to account for internal driving and combined with the Resistive Force Theory for hydrodynamic forces to compute the spatiotemporal power distribution of the internal forces exerted by protein motors in the sperm axoneme. Representative beat patterns and internal power distributions are computed for a large number of sperm samples from mutant mice deficient in a family of proteins that regulate calcium ion flux in the flagellum. Clear differences in beat patterns are observed which are found to be correlated with the active power distribution.
velocities of single cells and colonies for different prey sizes and initial positions to compare their feeding performance. The presence of background flow affects the sperm trajectory and hence the success rate of fertilization. We have studied the effect of unbounded simple shear flow and Poiseuille flow on the sperm trajectory. The sperm moves on an elliptical trajectory in the reference frame advecting with the local background flow in the simple shear flow and the length of the major-axis of this elliptical trajectory decreases with the shear rate. In the presence of Poiseuille flow, the sperm moves downstream or upstream depending on the flow strength. The sperm also moves toward the centerline in a Poiseuille flow. The cross-stream migration velocity of sperm decreases as the transverse distance of the sperm from the centerline decreases in the close vicinity of the centerline, while it increases far away from the centerline. We use sperm number, a dimensionless number representing the ratio of viscous force to elastic force, to study the effect of flagellar flexibility on the sperm trajectory. The length of the major axis of the elliptical trajectories increases with the sperm number in the simple shear flow and the cross-stream migration velocity of sperm increases with the sperm number in a Poiseuille flow.


d_1 National Science Foundation

10:10AM H31.00011 Finite Element Modeling of Microswimmers with Applications in Reproductive Biology, CARA NEAL, DAVID SMITH, MEURIG GALLAGHER, THOMAS MONTENEGRO-JOHNSON, University of Birmingham — The journey of human sperm to the egg is a complex and extremely important process. Sperm cells must navigate the intricate geometry of the fallopian tubes, generating active bending to swim through cervical mucus - a highly viscous, rheologically complex fluid. Many current models of sperm locomotion are computationally expensive, and most make the inaccurate assumption that the surrounding fluid is Newtonian. Here we develop a method capable of dealing with the non-linear equations associated with non-Newtonian fluids. This method uses a combination of a finite element technique and an elasto-hydrodynamic integral formulation (Hall-McNair et al. 2019) to model sperm cells with active flexible flagella. This formulation provides an efficient way of modeling single or multiple cells, accounting for the hydrodynamic interactions between them. In particular, the finite element component is formulated in such a way that the solution can be calculated on a coarse mesh, for reduced computational costs compared to more commonly used body-fitted meshes. We study how the model can be made more biologically accurate through the inclusion of varying bending stiffness as well as a passive distal region of the flagellum, and the subsequent effect on swimming efficiency.

10:23AM H31.00012 Interaction of spermatozoa with micro structured surfaces, VASILY KANTSLER, University of Warwick, ANTON BUKATIN, St. Petersburg Academic University, ENKELEIDA LUSHI, New Jersey Institute of Technology, PETER DENISSENKO, University of Warwick — Spermatozoa navigation plays crucial role in the process of mammalian fertilization. Mechanical interactions in the heterogeneous environment of the fertility such as surface scattering and rheotaxis are the key mechanisms in determining the sperm journey towards the ovum. Here we report an experimental study of interaction mechanisms for single human sperm cells scattering off solid surface boundaries of different curvature. The investigation is based on measuring the trajectories of the cells near convex and concave surfaces with 30 ~ 300 um radii of curvature in a microfluidic device. By analysing several thousands of cells’ trajectories we built the residence time dependences, concentration profiles the scattering distributions as a function of the surface curvature. For the concave objects, we have identified the surface curvature corresponding to the minimum of the cell residence time within the concave centerline. We use sperm number, a dimensionless number representing the ratio of viscous force to elastic force, to study the effect of flagellar flexibility on the sperm trajectory. The length of the major axis of the elliptical trajectories increases with the sperm number in the simple shear flow and the cross-stream migration velocity of sperm increases with the sperm number in a Poiseuille flow.

1 National Science Foundation

8:00AM H32.00001 Approach to finite-size prey by the choanoflagellate Salpingoeca rosetta, KIARASH SAMSAMI, HENRY FU, University of Utah — The choanoflagellate S. rosetta is a unicellular eukaryote with a single flagellum and a collar of microvilli. It feeds on bacteria by utilizing the fluid currents generated by the beating of its flagellum that bring its prey to the surface of the collar and ingesting the bacterium. S. rosetta is observed as a single thecate cell, a single swimming cell, a rosette shaped colony with flagella pointing outwards, and a chain shaped colony with each cell attached laterally to two neighboring cells. As a close relative of animals, this ability to form colonies and the possible resulting survival advantages could provide insight on origins of multicellularity. Prior works have studied fluid uptake and flow fields for S. rosetta single cells and colonies as a measure of feeding performance. However the bacterial prey is comparable in size to the S. rosetta cell and hence may have a considerable effect on the flow in near field. Here we study the hydrodynamics of a single cell or a colony approaching a finite size prey, simulating actual geometries of both. We study the approach time and velocities of single cells and colonies for different prey sizes and initial positions to compare their feeding performance.
8:13AM H32.0002 Bacterial Motility Near a Surface. NICHOLAS COLTHARP, Trinity University — Bacteria spend much of their time in complex environments: colonies of bacteria form biofilms on surfaces, and even free-swimming bacteria may find their range of motion limited by their environment. To understand how they navigate through such environments, our first step is to construct a physically-realistic model of an E. coli bacterium. We then use the method of regularized Stokeslets and the method of images to compute its swimming speed, body rotation rate, and flagellar torque in a homogeneous viscous fluid. As we vary the distance of the model bacterium from a surface, our results agree well with those of other techniques, and with no experimental values. We also simulate a bacterium swimming in heterogeneous fluids with suspended microstructures such as elastic polymers and filamentous networks, similar to what real bacteria experience.

1Trinity University and NSF grant DMS-1720323

8:26AM H32.0003 In silico micro-swimmers: runs and tumbles. SOOKKYUNG LIM, University of Cincinnati — We present a mathematical model of a micro-swimmer E. coli that can freely run by a flagellar bundle and tumble by motor reversals. The Kirchhoff rod theory is used to model the elastic helical flagella and the rod-shaped cell body is represented by a hollow ellipsoid that can translate and rotate as a neutrally buoyant rigid body. The hydrodynamic interaction between the fluid and the bacteria is described by the regularized version of Stokes flow. In this talk, we will focus on how bacteria can swim and reorient the course of swimming and how parts of bacteria play a role in swimming.

1NSF & Simons Foundation

8:39AM H32.0004 Swimming microalga as a micro-mixer in confined spaces. MOJTABA JARRAH, PEDRO ARANA-AGUDELO, Université Paris-Sud, LIMSI-CNRS, ADAMA CREPYY, Université Paris-Sud, FAST-CNRS, BEHNAM TAIIDI, CentraleSupélec - LGPM, HAROLD AURADOU, Université Paris-Sud, FAST-CNRS — The unicellular green alga Chlamydomonas uses two anterior flagella to swim in a breaststroke-like pulling motion, covering a distance of about seven to ten times its body size per second. This fast swimmer stirs the surrounding fluid like a mobile micro-mixer. How strong is this mixing? The few studies available on this subject, measured the diffusion of tracer particles around Chlamydomonas (a single cell or/and a suspension) to show mixing enhancement. However, we know that mixing process consists of folding and stretching of fluid elements. Diffusion of the tracer particles cannot be enough appropriate to characterize mixing features. In this work, we follow the separation of the tracers, initially close together, when swimming of a Chlamydomonas cell influences them. In this way, we quantify the mixing in different zones around the microswimmer. Some other aspects of the motility of Chlamydomonas in a confined environment, like residence time and light effects, are also investigated. Microscopy of Chlamydomonas and the tracers (Carboxylated yellow-green polystyrene microspheres, diameter = 0.5 μm) was carried out inside square bottom well arrays (125μm x 125μm x 60μm).

1This work was carried out with financial support of LaSIPS Paris-Saclay

8:52AM H32.0005 Squirmer in a density-stratified fluid. VASEEM SHAIK, AREZOO ARDEKANI, Purdue University — In this work, we analyze the motion of a two-mode spherical squirmer in a linearly density stratified fluid using the method of matched asymptotic expansions. We assume that the quasi-steady conditions prevail, inertia is negligible, stratification and advective transport rate of density are small, and the swimmer is oriented either vertically upwards or downwards. We consider a swimmer that is either far away from its neutrally buoyant position (NBP) or close to its NBP. Using the Boussinesq approximation, we find that stratification reduces the speed of a swimmer that is far from its NBP irrespective of the swimming gait while the stratification reduces (resp. increases) the speed of a pusher (resp. puller) swimmer that is close to its NBP. We can understand the former observation by considering a settling rigid sphere in a stratified fluid whereas the latter observation is consistent with the reported direct numerical simulations. Close to the swimmer, the flow field is approximately the same in the homogeneous and the stratified fluid but far from the swimmer, the flow field in these two fluids is significantly different. We also comment on the power consumption and the swimming efficiency in a stratified fluid.

9:05AM H32.0006 Fibrous flagellar hairs of Chlamydomonas reinhardtii do not enhance swimming. GUILLERMO AMADOR, DA WEI, MARIE-EVE AUBIN-TAM, DANIEL TAM, Delft University of Technology — The flagella of Chlamydomonas reinhardtii possess fibrous ultrastructures of nanometer-scale thickness known as mastigonemes. While these structures are hypothesized to enhance flagellar thrust, detailed hydrodynamic evidence supporting this claim is lacking. In this study, we present a comprehensive investigation into the hydrodynamic effects of mastigonemes using genetically modified mutants lacking the fibrous structures. Through high speed observations of freely swimming cells, we found the average and maximum swimming speeds to be unaffected by the presence of mastigonemes. In addition to swimming speeds, no significant difference was found for flagellar gait kinematics. Following our observations of swimming kinematics, we present direct measurements of the hydrodynamic forces generated by flagella with and without mastigonemes. These measurements were conducted using optical tweezers, which enabled high temporal and spatial resolution of hydrodynamic forces. Through our measurements, we found no significant difference in propulsive flows due to the presence of mastigonemes. Direct comparison between experimental measurements and numerical simulations revealed that swimming hydrodynamics were accurately captured without including mastigonemes on the modeled swimmer’s flagella. Therefore, mastigonemes do not appear to increase the flagella’s effective area while swimming, as previously thought. Our results refute the claim that mastigonemes enhance flagellar thrust in C. reinhardtii, and so their function still remains enigmatic.

9:18AM H32.0007 Hydrodynamic and chemotactic influences in bacterial foraging. NIKHIL DESAI, VASEEM SHAIK, AREZOO ARDEKANI, Purdue University — The discovery that marine bacteria can break down the hydrocarbons in oil for nutrition has motivated a fundamental understanding of the mechanisms underpinning oil-microbe interactions, e.g., the effect of oil-water interfaces on the hydrodynamics of swimming microbes. Hydrodynamic interactions enable the passive capture of microswimmers around rigid/fluid spherical obstacles, which has important consequences on a bacterium’s ability to populate nutrient sources like marine snow, oil drops etc. In this talk, we first demonstrate theoretically, that surfactant-laden drops act as more effective hydrodynamic traps for bacteria than clean drops. Next, we explain the important differences between hydrodynamic trapping around stationary versus translating obstacles. Finally, we show how hydrodynamic interactions are complemented by chemotaxis to non-trivially alter the distribution of marine bacteria around both stationary oil drops and sinking marine snow particles effusing hydrocarbon/nutrient plumes. We thus delineate the effects of various physicochemical influences—like nutrient distribution, fluid-flow and proximity to interfaces—on microbial behavior in natural environments.
9:31AM H32.00008 Hydrodynamic essentials of flagellar bundling. ALEXANDER CHAMOLLY, ERIC LAUGA, University of Cambridge — A lot of recent research activity has addressed the swimming dynamics of prokaryotic cells, in particular the formation of bundles of helical flagella in bacteria such as E. coli. The exact dynamics of this process are complex, and most hitherto proposed computational models aiming for a detailed description have involved an interplay between a number of physical effects including long-range hydrodynamic interactions, elastic restoring forces and short-range steric interactions while respecting the overall force and torque balance between flagella and the cell body. In this study we aim to understand what fundamental physical mechanisms trigger bundle formation in the first place. We distinguish between active bundling, induced by hydrodynamic interaction of the flexible flagella with each other, and passive bundling, triggered by advection of fluid around a moving cell body. We propose a minimal analytical model that involves only the essential hydrodynamics of flagellar propulsion and show that it is able to predict the dynamics of bundling, as well as the relative strength of both hydrodynamic effects.

9:44AM H32.00009 A minimal model for Spiroplasma chemotaxis1, CHRISTIAN ESPARZA LOPEZ, ERIC LAUGA, University of Cambridge — Spiroplasma is a small helical bacterium that swims and performs chemotaxis in a non conventional way. Instead of actuating flagella, it swims by progressively shifting the chirality of its body. The change in geometry gives rise to a wall domain - a kink - which propagates along the cell body. The chirality is then reverted in a similar fashion completing a swimming stroke. The whole deformation is non-reciprocal in time, therefore movement at low Reynolds number is achieved with the bacterium moving in the direction opposite to the kink pair propagation. Based on experimental observations, we develop a minimal model to describe Spiroplasma chemotaxis. We start by a simple resistive force theory model of the bacterium swimming gait. Using symmetry arguments we show how to calculate the net rotation and translation of the bacterium during one full stroke. We obtain expressions for the linear displacement as a function of the time between kinks, , that compare favourably with numerical computations. Using our theoretical results, we then construct a random walk model for Spiroplasma and we obtain expressions for its diffusivity as a function of and the kink angle .

1This project has received funding from the European Research Council (ERC) under the European Union’s Horizon 2020 research and innovation programme (grant agreement 682754 to EL).

9:57AM H32.00010 Hydrodynamics of bacterial spinning top1, KENTA ISHIMOTO, Kyoto University — We have theoretically and numerically investigated mono-flagellated bacterial swimming dynamics near a wall boundary with considering elastic hook flexibility to understand hydrodynamic interactions underlying the bacterial upright spinning motion, which has referred as low-Reynolds-number spinning top in a recent experimental study. We establish an elastohydrodynamic stability theory and found that the vertical spinning motion is enabled by the mechanical competition between the destabilization by the flagellar propulsion and rotational stabilization by the elastic coupling of the hook. These results demonstrate the mechanical nature of the behaviours in rich diversity and could contribute to our deeper understandings of the bacterial surface motility and biofilm formation.

1This work is supported in part by MEXT Leading Initiative for Excellent Young Researchers (LEADER), JSPS Grant-in-Aid for Young Scientists (18K13456), and JSPS Overseas Research Fellowship (29-0146).

10:10AM H32.00011 Hydrodynamics of prey capture in ciliated microorganisms1, MADIS RODE, Department of Physics and Centre for Ocean Life, Technical University of Denmark, THOMAS KIJORBOE, National Institute of Aquatic Resources and Centre for Ocean Life, Technical University of Denmark, ANDERS ANDERSEN, Department of Physics and Centre for Ocean Life, Technical University of Denmark — Unicellular microorganisms play a key role in the biological processes in the ocean. Here we focus on the marine ciliate Euplotes vannus, which uses complex arrangements of cilia with periodic beat patterns to generate a feeding flow, retain prey particles, and transport the retained particles to the mouth region of the cell. We describe the hydrodynamics of prey capture to answer how the flow-rate of the feeding current and the prey size spectrum depend on motion and design of the organelles. We use particle image velocimetry to determine the feeding flow quantitatively, and particle tracking to identify retained and lost prey particles and to follow their transport to the mouth region. We have observed E. vannus to both swim freely and crawl or sit on solid surfaces. When freely swimming, the ciliate generates a puller-like flow field seemingly without feeding, and when sitting it generates a strong feeding flow that resembles the flow due to a point force at the front of the cell.

1The project is supported by The Independent Research Fund Denmark (grant no. 7014-00033B), and the Centre for Ocean Life, a VKR Centre of Excellence supported by the Villum Foundation.

10:23AM H32.00012 Dynamics of growth and form in prebiotic vesicles, THOMAS FAI, Brandeis University, TERESA RUIZ-HERRERO, L. MAHADEVAN, Harvard University — The growth, form, and division of prebiotic vesicles, membraneous bags of fluid of varying components and shapes is hypothesized to have served as the substrate for the origin of life. The dynamics of these out-of-equilibrium structures is controlled by physicochemical processes that include the intercalation of amphiphiles into the membrane, fluid flow across the membrane, and elastic deformations of the membrane. To understand prebiotic vesicular forms and their dynamics, we construct a minimal model that couples membrane growth, deformation, and fluid permeation, ultimately coupled in terms of two dimensionless parameters that characterize the relative rate of membrane growth and the membrane permeability. Numerical simulations show that our model captures the morphological diversity seen in extant precursor mimics of cellular life, and might provide simple guidelines for the synthesis of these complex shapes from simple ingredients.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H33 Flow Instability: Nonlinear Dynamics 615 - Thomas Ward, Iowa State University

8:00AM H33.00011 Linear stability analysis of a miscible two-layer power-law fluid in Couette flow1, THOMAS WARD, TEJASWI SOORI, Iowa State University — We examine the stability of a miscible two-layer power-law fluid in a 2D shear flow. The problem is motivated by recent experiments showing that miscible displacement of shear-thinning fluids yields an instability. To study fluid stability we analyze the mass, momentum and species conservation equations in a Couette flow geometry. Layer depth, consistency and power-law index are varied for zero wall slip conditions. We also examine the interface by treating the two-layer problem as a two-fluid one with zero surface tension. The governing equations are linearized about base states to estimate growth rates for dependent variables in the limit of infinitesimally small disturbances. We integrate the resulting ODE equations using a standard Chebyshev collocation method. We also compare results with prior ones for two-fluid layer and miscible two-layer problems, along with critical values observed in experiments.

1 ACS-PRF
8:13AM H33.00002 Exploiting noise-induced dynamics for system identification near a Hopf bifurcation

MINWOO LEE, The Hong Kong University of Science and Technology, YUANHANG ZHU, Brown University, YU GUAN, LARRY K.B. LI, The Hong Kong University of Science and Technology — We propose a system identification framework that exploits the noise-induced dynamics inherent to nonlinear systems near a supercritical or subcritical Hopf bifurcation. The key assumption made is that the system response can be modeled with a Stuart–Landau equation and its corresponding Fokker–Planck equation. We demonstrate the framework on two different flow systems: a hydrodynamic system (a low-density jet) undergoing a subcritical Hopf bifurcation, and a thermoacoustic system (a Rijke tube) undergoing a supercritical Hopf bifurcation. For both systems, we extract the model coefficients using experimental measurements of the noise-induced dynamics in only the unconditionally stable regime, prior to both the Hopf and saddle-node points. We show that the framework can accurately predict (i) the order of nonlinearity, (ii) the types and locations of the bifurcation points, and (iii) the limit-cycle characteristics beyond such points. As the noise-induced dynamics of nonlinear systems are expected to be universal in the vicinity of a Hopf bifurcation (Ushakov et al. 2005, Phys. Rev. Lett., vol. 95, 123903), the proposed framework should be applicable to a variety of flow systems in nature and engineering. 

1This work was funded by the Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815).

8:26AM H33.00003 The Role of Cubic Nonlinearity in Limit Cycle Oscillations of Variable-Density Shear Flows

CHRISTOPHER DOUGLAS, BENJAMIN EMERSON, TIMOTHY LIEUWEN, Georgia Institute of Technology — This work is directed toward understanding the limit cycle features of globally unstable flows with variable density. Analysis of limit cycles in constant-density shear flows have revealed that oscillation characteristics are controlled by two key effects: mean flow distortions and harmonic interactions. Both of these effects are manifestations of the quadratic nonlinearity of the incompressible Navier-Stokes equations, and their respective roles vary in different flow configurations. However, in flows such as bluff body wakes, surprisingly accurate predictions of important limit cycle features such as frequency and amplitude are possible by neglecting the latter effect and performing a linear analysis about the mean flow. Crucially, even when its amplitude is not small, the quadratic nonlinearity does not allow the limit cycle fundamental to directly modify itself. Conversely, when the fluid density is not constant, a cubic nonlinearity arises and enables self-interactions which are independent from the other effects. This calls into question the validity of linear approaches in such contexts and motivates an investigation of the role of these triadic self-interactions in variable-density shear flows.

1This research was supported by the University Turbine Systems Research program, contract #DE-FE0031285 (contract monitor Dr. Mark Freeman).

8:39AM H33.00004 Nonlinear Fluid Flow Analysis Using Integral Quadratic Constraints

ANIKETH KALUR, PETER J. SEILER, MAZIAR S. HEMATI, University of Minnesota, Twin Cities — The exact mechanism for sub-critical transition to turbulence in shear flows is complicated and not fully understood due to the interaction between the linear and nonlinear terms in the Navier-Stokes equations (NSE). The linear operator in NSE causes a transient amplification of perturbation energy – a necessary condition for sub-critical transition. The nonlinearity in NSE acts in feedback with the linear system and mixes energy between modes. This static lossless nonlinearity is responsible for triggering transition and sustaining turbulence. In this talk, we will show that the nonlinearity in NSE can be replaced by a set of integral quadratic constraints (IQCs), which effectively represent correlations between the inputs and outputs of the nonlinearity. Thus, analysis of nonlinear flows can be cast as problems in the analysis of linear dynamics – coupling energy-based methods with corresponding IQCs. We perform our investigations on the Waleffe-Kim-Hamilton (WKH) model, which is a low-dimensional mechanistic model developed to capture the physics of transition. IQC-based stability and performance analysis of the WKH model will be presented.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-19-1-0034, monitored by Dr. Gregg Abate.

8:52AM H33.00005 Effects of entry shapes on evolution and transition mechanisms of internal swirling flows

XINGJIAN WANG, Florida Institute of Technology, YANXING WANG, New Mexico State University — Previous works have investigated the characteristics of the central recirculation zone and intrinsic instability waves of a swirling flow injected through a tangential slit entry. In practice, orifice entry is frequently used to generate the swirling motion in a cylindrical chamber, but is much less documented. In this study, we numerically explore flow evolution and transition mechanisms of internal swirling flow with orifice entry using Galerkin finite element method. A grid convergence study is conducted to ensure the appropriate grid resolution at the orifice entry and in regions with complex flow structures. The effects of Reynolds number and swirl level controlled by the orifice angle are examined in detail. The numerical results of tangential slit and orifice entries will be compared systematically in terms of flow topologies and underlying instability mechanisms. A unified theory connecting different flow states of swirling flow will be established.

9:05AM H33.00006 On slow–fast generalized quasilinear dynamical systems

GREG CHINI, University of New Hampshire, ALESSIA FERRARO, Swiss Federal Institute of Technology Lausanne (EPFL), GUILLAUME MICHEL, Ecole Normale Superieure, CNRS, COLM-CILLE CAULFIELD, Cambridge University (DAMTP) — The quasilinear (QL) reduction has proved surprising useful in the analysis and modeling of anisotropic turbulent flows. As first introduced in the context of fluid turbulence by Stuart (1958), the QL approximation involves parsing flow variables into suitable (e.g., streamwise) mean and fluctuation fields and then retaining fluctuation-fluctuation nonlinearities only where they feed back onto the mean fields through Reynolds stress divergences. Although increasingly invoked as a useful albeit ad hoc approximation, the QL reduction can be formally justified in the asymptotic limit of temporal scale separation between the mean and fluctuation dynamics, as arises, e.g., in the asymptotic description of strongly stratified shear turbulence and of certain exact coherent states in wall-bounded shear flows. A fundamental and vexing feature of such systems, however, is that when the slow mean fields are locally frozen in time, the fast linearized dynamics can admit exponential fluctuation growth. In this work, a new multiscale formalism is introduced that obviates the need to co-evolve the mean and fluctuation dynamics on the fast time scale (i.e., the usual fix) by exploiting the necessity of slow–fast QL systems to self-tune toward a marginal-stability manifold.
9:18AM H33.00007 Stability of Flow past a Freely-Rotatable Sprung Cylinder. KE DING, ARNE PEARLSTEIN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Recent work (Tumkur et al., J. Fluid Mech. 828, 196-235, 2017; Blanchard et al., Phys. Rev. Fluids, 4, 054501, 2019) considers flow past a linearly-sprung circular cylinder whose transverse rectilinear motion is inertially coupled to linearly-damped rotational motion of an attached mass about the cylinder axis. (The rotating mass is either inside the cylinder or beyond the span of the flow, thus having no contact with the fluid.) That work reveals chaotic response at Reynolds numbers (Re) well below the fixed-cylinder critical value $Re_{fixed,c}$ and multiple unsteady long-time solutions for many combinations of the parameters. Here, we consider flow past a linearly-sprung circular cylinder with a nonaxisymmetric density distribution, which is free to rotate, and whose rotational motion is linearly damped. As for the case where the cylinder cannot rotate, a steady, symmetric, motionless-cylinder (SSM) solution exists for all values of the parameters and all density distributions and orientations. We use a spectral-element technique to investigate the stability boundary in the space of Re and spring stiffness, and find that over a limited stiffness range, rotatability renders the SSM solution stable for Re >$Re_{fixed,c}$. We also show that stability boundaries for different (initial) orientations of the density distribution are very similar to those for different (initial) positions of the attached mass, suggesting that much of the complex dynamical behavior in the attached-mass case can be realized using a cylinder with nonuniform density, with no attached mass.

9:31AM H33.00008 Effect of electric field on the linear stability characteristics of two-layer channel of Newtonian and Herschel-Bulkley fluids. GAUTAM KUMAR, PURANAM ANANTH L NARAYANA, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We investigate the effect of electric field on the linear stability characteristics of pressure-driven flow in a channel, wherein a Newtonian fluid layer superposed on a layer of Herschel-Bulkley fluid. Both fluids are assumed to be incompressible and leaky dielectric media. The modified Orr-Sommerfeld eigenvalue problems are derived and solved using an efficient spectral collocation method. An asymptotic analysis is also performed in the long-wave limit. The effects of electric field, Bingham number, flow index and the ratios of density, viscosity, electrical conductivity and permittivity between the fluids are studied. We observed that the electric field can stabilize or destabilize the systems in different regimes.

9:44AM H33.00009 Genesis of Taylor–Couette Flow Instabilities, H. OUALLI, M. MEKADEM, M. KHIENNASS, Y. REZGA, S. TEBTAB, T. AZZAM, Ecole Militaire Polytechnique, Algiers, Algeria, A. BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria, M. GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Numerical simulations are conducted of a Taylor–Couette flow from early structuring stages to completion of the Taylors axial stationary waves. We seek to elucidate the underlying mechanisms responsible for the genesis of this flow type and to identify the intermediate embryonic staged growth and completion of the Taylor and Couette rolls and cylinders are implemented on FLUENT. The calculations are based on the finite-volume method with a mesh size of 32×28×256 in, respectively, the radial, azimuthal, and axial directions. The simulations are validated using prior experimental results. The calculations span Taylor numbers from $Ta = 10^9$ to $Ta = 43.8$. The results show that the incipient pressure variations are of the order of $10^{12}$ Pa, detected at $Ta = 10^3$, on four symmetrically separated cardinal points within the system. When $Ta > 10^4$, a progressive propagation of alternating overpressure and depression zones operate in both azimuthal directions. This is the first step in the chain of mechanisms responsible for the Taylors wave building process. The study reports, for the first time, all the details to explain the instability mechanisms evolution.

9:57AM H33.00010 Linear and nonlinear thermosolutal instabilities in an inclined porous layer. PURANAM ANANTH L NARAYANA, GAUTAM KUMAR, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We investigate the double-diffusive instability in an inclined porous layer with a concentration based internal heat source by performing linear instability and nonlinear energy analyses. The effects of various dimensionless parameters, such as the thermal ($Ra_v$) and solutal ($Ra_c$) Rayleigh numbers, the angle of inclination ($\phi$), the Lewis number ($Le$), and the concentration based internal heat source ($Q$) have been investigated. A comparison between the linear and nonlinear thresholds for the longitudinal and transverse rolls provides the region of the subcritical instability. We found that the system becomes more unstable when the diffusivity of the temperature is larger than that of the solute and with an increase in the source strength. It is observed that increasing inclination angle stabilizes the system. Although the longitudinal roll remains stationary without the region of subcritical instability, the transverse roll transforms changes from stationary-oscillatory-stationary mode with the increase in the inclination angle.

10:10AM H33.00011 Stability of the interface of an isotropic active fluid1, WAN LUO, HARSH SONI, ROBERT PELCOVITS, THOMAS POWERS, Brown University — We study the linear stability of an isotropic active fluid in three geometries: a film of active fluid on a rigid substrate, a cylindrical thread of fluid, and a spherical fluid droplet. The active fluid is modeled by the hydrodynamic theory of an active nematic liquid crystal in the isotropic phase. In each geometry, we calculate the growth rate of sinusoidal modes of deformation of the interface. There are two distinct branches of growth rates: at long wavelength, one corresponds to the deformation of the surface, and one corresponds to the evolution of the liquid crystalline degrees of freedom. The passive cases of the film and the droplet are always stable. For these geometries, a sufficiently large activity leads to instability. Activity also leads to propagating damped or growing modes. The passive cylindrical thread is unstable for perturbations with wavelength longer than the circumference. A sufficiently large activity can make any wavelength unstable, and again leads to propagating damped or growing modes. Our calculations are carried out for the case of zero Frank elasticity. While Frank elasticity is a stabilizing mechanism as it penalizes distortions of the order parameter tensor, we show that it has a small effect on the instabilities considered here.

10:23AM H33.00012 The stability of evaporating binary liquid film heated from below1, ROBSON NAZARETH, School of Engineering, University of Edinburgh, GEORGE KARAPETAS, Department of Chemical Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, School of Engineering, Aristotle University of Thessaloniki, PEDRO SAENZ, Department of Mathematics, Massachusetts Institute of Technology, OMAR MATAR, Department of Chemical Engineering, Imperial College London, KHELLIL SEFIANE, P
The liquid water contains Sodium Fluorescein in concentrations below 168 ppm. Icicles exhibit ripples around their circumference with a near universal wavelength of 1 cm. Experiments have shown that the rippling instability is associated with small levels of impurities and is not present for sufficiently pure water. Existing models of icicle growth assume that the icicle is covered completely by a thin film of flowing water. In our work, we consider perturbations around this base state and employ Floquet-Bloch theory to account for disturbances of arbitrary wavelengths, i.e. not necessarily matching the periodicity of the substrate. Through numerical simulations, we highlight the effect of inertia, viscous, and capillary forces on the stability of the fluid flow and also examine in detail the effect of substrate wettability, orientation with respect to gravity and geometric characteristics of the substrate. Moreover, the existence of the freely moving contact lines inside the cavity gives rise to multiple steady states which are analyzed for their stability. The role of air inclusions in the stabilization of the liquid film and the related mechanism will be discussed.

The authors acknowledge the financial support from the LIMMAT foundation under the grant MuSiComPS and from the Hellenic Foundation for Research and Innovation (HFRI) and the General Secretariat for Research and Technology (GSRT), under grant agreements No 1743 and No 792.

We conclude that the roll-wave instability is analogous to the critical point of the roll-wave instability in a fluid film at Fr ≈ 1. We conclude that the roll-wave instability of the gravity current is responsible for the texture in a glass of Guinness beer.
9:18AM H34.00007 Bistability in the Flow of Polymer Solutions in Porous Media

CHRISTOPHER BROWNE, AUDREY SHIH, SUJIT DATTA, Department of Chemical and Biological Engineering, Princeton University — Polymer solutions are often injected in porous media to improve oil recovery or groundwater remediation, but applications are limited by an incomplete understanding of the underlying physics. In a tortuous pore space, the flow becomes unstable at sufficiently large injection rates. However, how the spatio-temporal characteristics of this flow state depend on pore geometry is poorly understood. We shed light on this question by systematically varying the spacing between pores. When the pore spacing is large, unstable eddies form upstream of each pore, similar to the case of an isolated pore. By contrast, when the pore spacing is sufficiently small, the flow exhibits a surprising bistability, stochastically switching between two distinct flow states. We hypothesize that this unusual behavior arises from the interplay between the retention of polymer strain between pores, hysteresis in polymer conformations, and fluctuations in the flow. Consistent with this idea, the mean flow state can be tuned by the imposed flow rate. Moreover, we find that while flow state is correlated between neighboring pores, these correlations do not persist long-range. Our results thus help to elucidate the rich array of flow behaviors that can arise in polymer solution flow through porous media.

9:31AM H34.00008 Elastic and Shear thinning instabilities for the flow of polymer solutions through microribbons

BIKHAN CHANDRA, VISWANATHAN SHANKAR, Indian Institute of Technology Kanpur — It is well known that the flow of dilute polymer solution through tubes undergoes a non-linear subcritical transition at Re slightly higher than 2000. However, when the polymer concentration is increased beyond a threshold concentration, the instability occurs at a Re much lower than 2000. The instability occurring at Re lower than 2000 at higher polymer concentration (hence higher elasticity) is a linear instability. A recent article by Poole, 2016, shows that if the polymer solution prepared is sufficiently shear thinning in nature, an instability occurs at very low inertia (Re<50). We explore the possibility of very low Re instability for the flow of concentrated polymer solutions through micro-tubes. High concentration of polymer solution coupled with small tube diameters enable us to reach high elasticity numbers. We observe that the polymer solution destabilize at very low Re (Re<10). The nature of instability is also observed to be very different as compared to elastio-inertial instability as observed from the scaling relationships.

9:44AM H34.00009 Soft Cell: Flow-induced deformation of a compliant Hele-Shaw cell

FINN BOX, School of Physics, University of Manchester, GUNNAR PENG, DAMTP, University of Cambridge, ANNE JUEL, DRAGA PIHLER-PUZOVIC, School of Physics, University of Manchester — We present an experimental study of the flow-induced deformation of a compliant Hele-Shaw cell, comprising a soft substrate and a rigid upper boundary separated by a thin gap. An axisymmetric displacement flow is formed by injecting viscous fluid, at a constant volumetric flux, into the center of the cell, which is pre-filled with the same fluid. By measuring the surface deformation of the substrate, we find that the deformation profile is self-similar during a period of transient growth before rapidly attaining a steady shape determined by the logarithmic profile of the fluid pressure within the cell. We discuss how the deformed substrate influences the motion of the advancing front between injected and resident file in the cell.

9:57AM H34.00010 Control of Radial Miscible Viscous Fingering Using a Finite Blob-An Experimental Study

SADA NAND, VANDITA SHARMA, MANORANJAN MISHRA, Indian Institute of Technology Ropar — We experimentally demonstrate the control of miscible viscous fingering instability in a novel way. We consider the less viscous fluid initially in a circular blob of finite radius r0 displacing the surrounding more viscous fluid in the radial Hele-Shaw cell. Experiments only with a point source (r0=0) are available in the literature. Getting initial finite circular blob is a huge experimental challenge. For each r0, a flow rate is wisely calculated so as to have a stable displacement up to radius r0. Also, a diligently designed T- junction is utilized in the Hele-Shaw experimental set-up to ease the fluid injection. The experiments depict the initial radius of the circular blob (r0) as a controlling parameter. A delay in instability is observed for experiments with non-zero r0 in comparison to those performed with a point source. Further, a reduced instability is evident with an increase in r0, which is in agreement with the numerical simulations performed.

Monday, November 25, 2019 8:00AM - 10:36AM
Session H35 Microscale Flows: Interfaces and Wetting 617 - John Sader, The University of Melbourne

8:00AM H35.00001 DEM Modeling of Coupled Multiphase Flow and Granular Mechanics: Wettability Control on Fracture Patterns

YUE MENG, BAUYRZHAN PRIMKULOV, Massachusetts Institute of Technology, ZHIHONG YANG, State Key Laboratory of Water Resources and Hydropower Engineering, Wuhan University, FIONA KWOK, University of Hong Kong, RUBEN JUANES, Massachusetts Institute of Technology — The interplay between multiphase flow in a granular medium and the displacement of the grain particles generates a wide range of patterns. The balance between frictional, viscous, and capillary forces has been studied in experiments and simulations, and has helped understanding the underlying mechanisms for a wide range of phenomena, such as methane migration in lake sediments. Here we study fluid-induced fracturing of granular media by hydromechanically coupling a moving capacitor dynamic network model with discrete element modeling. We inject a less viscous fluid into a frictional granular pack initially saturated with a more viscous, immiscible fluid under low capillary number. We study the impact of contact angle and initial packing density, and find four different regimes of the fluid invasion morphology: capillary invasion and fracturing, frictional fingers, capillary invasion, and capillary compaction. We rationalize these simulation outputs by means of a jamming analysis, which allows us to explain fracture initiation as emerging from a jamming transition. We synthesize the analysis in the form of a novel phase diagram of jamming for wet granular media.

8:13AM H35.00002 How the natural structure of cuttlebone facilitates efficient microfluidic transport

KARL FROHLICH, EHSAN EMSAILI, Cornell University, TING YANG, LING LI, Virginia Tech, SUNGHWAN JUNG, Cornell University — When a viscous fluid is displacing a less viscous fluid in a thin channel, the interface moves unevenly due to the Saffman-Taylor instability. Supressing such instability plays a key role in many petroleum industries and biological systems. One natural example of this phenomenon is in the bones of cuttlefish. These cuttlebones contain multiple parallel microscopic chambers which are reinforced by winding vertical wall-like structures. It has been studied that the cuttlefish regulates its buoyancy by pumping water in and out of this structure. In this study, we investigate the cuttlebone structure for its unique ability to suppress the instability of a moving liquid-gas interface. Cuttlebone samples were studied in vitro to understand how fluid moves through the structure at different pressures and flow rates. Additionally by using cuttlebone structure as biological inspiration, we fabricated simplified channels using 3D resin printing in order to more rigorously test the impact of the liquid-gas meniscus at varying channel curvatures, pressures and other arrangements. Results show that a presence in channel curvature and positioning, like that seen in cuttlebone, can prevent the uneven growth of menisci, helping to transport a liquid-gas interface more uniformly.
comparable to phase change rate. However, rate-dependent surface elasticity and surface viscosity emerge when the compression rate is
surfactants. Notably, our model predicts that liquid crystalline domains grow freely and the surfactant exerts no additional surface stresses
dynamic ‘freezing’ and ‘melting’ of insoluble surfactant domains as a 2D analog of classic works on dynamic adsorption and desorption of soluble
rheological stresses. Additional complexities often arise when insoluble surfactants exhibit non-trivial morphologies, for example, forming con-
flows involving fluid interfaces. Beyond reducing surface tension, surfactants may modify fluid flow by exerting Marangoni and/or surface
Barbara — Surfactants – molecules and particles that preferentially adsorb to fluid interfaces – play a ubiquitous role in industrial and biological
Tanner’s law suggested for spreading of a Newtonian fluid on solid substrates. We propose a scaling analysis based on a combination of the
extent to which, the spreading dynamics of the fluid contact-line on the solid substrate can affect the thinning dynamics of the fluid filament
formed between the needle and substrate. Experiments are performed using two flat substrates; a big substrate, where fluid contact-line in free
to move and a finite size substrate, where fluid contact-line is pinned. Two distinct flow regimes are observed. In regime I, the fluid wetting
dynamics does not significantly affect the filament thinning process. However, in regime II, the fluid wetting on the big solid substrate impacts
the filament thinning dynamics significantly by lowering the filament life time, extensional relaxation times and the Trouton ratios compared
to that of the pinned contact-line. Our analysis shows that spreading of these viscoelastic surfactant fluids are reasonably well captured by
Tanner’s law suggested for spreading of a Newtonian fluid on solid substrates. We propose a scaling analysis based on a combination of the
wetting forces and viscous dissipation that can successfully distinguish these two flow regimes from each other.

8:39AM H35.00004 Surface dilatational rheology during phase coexistence of insol-
uble surfactants , HARIHANKAR MANIKANTAN, University of California, Davis, TODD SQUIRES, University of California, Santa
Barbara — Surfactants – molecules and particles that preferentially adsorb to fluid interfaces – play a ubiquitous role in industrial and biological
flows involving fluid interfaces. Beyond reducing surface tension, surfactants may modify fluid flow by exerting Marangoni and/or surface
rheological stresses. Additional complexities often arise when insoluble surfactants exhibit non-trivial morphologies, for example, forming con-
densed phases with liquid crystalline order that coexist with disordered phases. Here, we show that the dynamic growth of liquid crystalline
domains at the expense of the continuous phase upon compression gives rise to a frequency-dependent viscoelastic behavior. We treat the
dynamic ‘freezing’ and ‘melting’ of insoluble surfactant domains as a 2D analog of classic works on dynamic adsorption and desorption of soluble
surfactants. Notably, our model predicts that liquid crystalline domains grow freely and the surfactant exerts no additional surface stresses
upon asymptotically slow compression. However, rate-dependent surface elasticity and surface viscosity emerge when the compression rate is
comparable to phase change rate.

8:52AM H35.00005 ABSTRACT WITHDRAWN —

9:05AM H35.00006 Size dependent droplet interfacial tension and surfactant trans-
port in oily bilgewater systems1 , YUN CHEN, CARI DUTCHER, University of Minnesota — Many liquid-liquid emulsions,
including shipboard oil bilge waters, are chemically stabilized by surfactants and additives. The emulsion stability is determined by the interfacial
tension (IFT) of surfactant-laden interface between the continuous and dispersed phase, as well as the size of the dispersed droplets. In the
present work, the dynamic IFT of droplets at micron-scale (~80 um) and milli-scale (~2 mm) is measured with simulated bilge waters with
soluble surfactant systems. It is found that the IFT of micron-scale droplets decays faster than that of the milli-scale droplets due to smaller
diffusion boundary layer thickness. When surfactants are added into aqueous phase for both water-in-oil and oil-in-water condition, the IFT of
micron-scale droplets decays faster when the surfactant is in outer phase than in the inner phase. In contrast, for the larger milli-scale droplets,
the IFT decay rate does not depend on which phase the surfactant is added. The observations are explained by the change in diffusion limited
to kinetic limited surfactant transport as the droplet size decreases. In addition, experimental results of droplet coalescence depending on the
IFT using model system is also presented in this work.

1SERDP project WP18-1031

9:18AM H35.00007 Characteristics of wetting distance in multilayered paper-based
channel1 , HYUNWOONG KANG, Dept. of Mechanical Engineering, Hanyang University, ILHOON JANG, Institute of Nano Science and
Technology, Hanyang University, SIMON SONG, Dept. of Mechanical Engineering, Hanyang University — A multilayered paper-based device
which has a gap between paper layers can generate much greater flow velocity than a typical single layer paper-based device. This feature
enables quick analysis in applications utilizing paper-based devices. Predicting the time to reach the detection point and the amount of sample
fluid is critical to improving detection accuracy and manufacturing the multi-functional device. However, the multilayered paper-based device
has a more complicated fluid flow than the conventional device resulting in the different flow with the single-layer device that can be described
by the Lucas-Washburn equation. These are due to the simultaneous sample flow flowing through both gap and paper layer. For example, since
the paper internal flow is much slower than the gap internal flow, the fluid can be absorbed into the paper layer in the thickness direction, and
it reduces a flow rate, especially at the initial stage of the fluid flow. In this study, we analyze the characteristics of the flow mainly focused
on the differences between the existing capillary driven flow model and explain the factors that affect the wetting distance in the multilayer
paper-based channel.

1This work was supported by the Human Resources Program in Energy Technology of the Korea Institute of Energy Technology Eval-
uation and Planning (KETEP), granted financial resource from the Ministry of Trade, Industry & Energy, Republic of Korea (No.
20184010201710)

9:31AM H35.00008 Flow topology and bifurcations in streaming lattices1 , YASHRAJ
BHOSALE, TEJASWIN PARTHASARATHY, GIRIDAR VISHWANATHAN, GABRIEL JUAREZ, MATTIA GAZZOLA, University of Illinois,
Urbana-Champaign — Viscous streaming flows generated due to constant curvature objects (circular cylinders, infinite plates) have been well-
understood in the past. Yet, characterization and understanding of such flows when multiple body length-scales are involved has not been
looked into. We propose a simplified setting to understand and explore the effect of multiple curvatures on streaming flows, analysing the
system through the lens of bifurcation theory. This analysis uncovers the dynamic richness of the system, which we showcase through potential
microfluidic applications.

1NSF CAREER Grant No. CBET-1846752 (MG)
9:44 AM H35.00009 Thermal behavior of slip length in connection with fluid-solid chemical affinity. ADRIANO GRIGOLO, THIAGO VISCONDI, JULIO MENEGHINI, Polytechnic School, University of São Paulo, IBERÊ CALDAS, Institute of Physics, University of São Paulo — The relation between fluid slip on a solid surface and flow temperature is investigated with the help of molecular dynamics simulations. An atomistic Couette configuration, consisting of a Lennard-Jones fluid and two rigid walls, is prepared and evolved to its steady state, where the slip length can be readily extracted from the linear velocity profile. Simulations are carried out for a broad range of temperatures and for different magnitudes of the fluid-solid chemical affinity. This last parameter is found to affect the overall shape of the slip length vs. temperature curve. Depending on its value, three distinct types of thermal behavior are observed: slips (unincreasing and decreasing), critical (constant), and sticky (increasing). We find that the temperature dependence of the slip length in each of these cases is well described by a simple heuristic function, whose adjustable parameters have a clear physical interpretation. The present study is aimed at the development of material-specific boundary conditions and their use in multiscale simulations. In this context, our analysis, once extended to more complex substances, may aid in the design and optimization of microfluidic devices and nanostructured membranes set to operate at specific temperatures.

The authors acknowledge the sponsorship of the Research Centre for Gas Innovation (FAPESP Proc. 2014/50.279-4) and SHELL Brasil.

9:57 AM H35.00010 Existence of the Navier slip condition for liquids. JESSE COLLINS, The University of Melbourne, SELIM OLCUM, Massachusetts Institute of Technology, DEBADI CHAKRABORTY, The University of Melbourne, SCOTT MANALIS, Massachusetts Institute of Technology, JOHN SADER, The University of Melbourne — The Navier slip condition is regularly used to characterize the interaction of a fluid with a solid boundary. Its use at the gas-solid boundary is justified rigorously from Boltzmann’s kinetic theory of gases, however no such parallel exists at the liquid-solid boundary. The strongest evidence for existence of the Navier slip condition at the liquid-solid interface comes from molecular dynamics simulations. As dictated by kinetic theory, the Navier slip length is a constitutive property that holds when the flow is continuum away from the solid interface. Here, we present an experimental protocol that is used to measure the Navier slip length on individual and isolated particles with exquisite precision. Experiments consisting of thousands of observations on individual gold nanoparticles give a constant slip length of $2.7\pm0.6$ nm — independent of particle size — providing experimental validation of the Navier slip condition for liquids.

10:10 AM H35.00011 Surface acoustic wave driven wetting transition. SUDEEPTHI A, ASHIS KUMAR SEN, Indian Institute of Technology Madras, LESLIE YEO, RMIT University, Australia — Water drop deposited on a nanostructured superhydrophobic surface undergoes wetting transition from Cassie-Baxter state to Wenzel state upon excitation by MHz frequency acoustic waves (surface acoustic waves). The wetting transition is governed by the competition between the input acoustic energy supplied to the drop via the surface acoustic waves (SAWs) and the energy barrier associated with the transition. The kinetic energy gained by the drop due to the transfer of energy from SAWs drive the liquid into the hydrophobic grooves to overcome the energy barrier associated with the replacement of low energy solid-gas interface with the high energy solid-liquid interface. As the liquid comes in contact with the substrate due to the filling of nanogrooves, boundary layer acoustic streaming assisted by the capillary flow in the hydrophilic bottom walled nanogrooves displaces the drop contact line giving rise to drop spreading. The transition is irreversible since the drop attains minimum energy state in the Wenzel condition.

10:23 AM H35.00012 In Situ Identification of Dewetting-Induced Large-Scale Deformation of Vertically Aligned Single-Walled Carbon Nanotubes. YUTA YOSHIMOTO, KOICHI ISOMURA, SOU SUGIYAMA, HUA AN, Department of Mechanical Engineering, The University of Tokyo, TAKUMA HORI, Department of Mechanical Engineering, Tokyo University of Science, TAIKI INOUE, SHOHEI CHIASHI, SHU TAKAGI, SHIGEO MARUYAMA, IKUYA KINEFUCHI, Department of Mechanical Engineering, The University of Tokyo — We investigated dynamical processes of capillary-mediated deformation of vertically aligned single-walled carbon nanotubes (VA-SWCNTs) via in situ observation of their wetting and dewetting behaviors using an environmental scanning electron microscope. Three types of wetting behaviors on a VA-SWCNT sample were observed, namely conical shaped water aggregates, spherical droplets on tips of conical shaped water aggregates, and extensively distributed water layers. The former two types both resulted in dimples on the VA-SWCNT surface, failing to induce large-scale deformation of VA-SWCNTs. In contrast, the latter caused the formation of wall-like structures and crack propagation in the VA-SWCNT film during the dewetting process due to directional retraction of vapor-liquid interfaces. Compared to the previous studies based on ex situ observations of dried samples, our in situ observation successfully captured temporal evolution of the dewetting-induced deformation, which represents initial stages of capillary processes that lead to the self-organization of VA-SWCNTs reported in recent literatures.

The authors acknowledge the sponsorship of the Research Centre for Gas Innovation (FAPESP Proc. 2014/50.279-4) and SHELL Brasil.

Monday, November 25, 2019 8:00 AM - 10:23 AM -
Session H36 Microscale Flows: Particles 618 - Michael Loewenberg, Yale University

8:00 AM H36.00001 Fiber buckling in confined viscous flows. ANKE LINDNER, JEAN CAPPELLO, OLIVIA DU ROURE, PMMH-ESPCI, PSL University, CNRS, Sorbonne University, Paris University, France, CAMILLE DUPRAT, Ladhyx, Polytechnique, France, MATHIAS Bichert, FRANÇOIS GALLAIRE, EPFL, Lausanne, Switzerland — Studying fluid structure interactions in confined flows is important to understand locomotion of micro-organisms in soils or medical conduct as well as the movement of long fibers in fractures, where they are used as in-situ probes for example in oil recovery. Here we look at the dynamics of a model system, constituted by flexible fibers freely transported in pressure-driven flows in a Hele-Shaw cell. Fiber height is comparable to channel height and in this very confined geometry the fiber dynamics are dominated by viscous friction with top and bottom walls. We focus on the dynamics of fibers oriented parallel to the flow direction and show that a buckling instability occurs under certain conditions. The fibers deform into a wavy shape resembling a wave-packet, with a well-defined dominant wavelength. Such an instability is triggered by a competition between viscous forces and elasticity and is observed only for long fibers, at least one order of magnitude larger than the observed wavelength. We characterize experimentally the instability and show that the wavelength of the deformation is proportional to an elasto-viscous length. We furthermore study the growth rate of the instability for different fiber geometries, flow strength, and mechanical properties of the fibers.

1 ERC Consolidator Grant no 682367
8:13AM H36.00002 Colloidal particle migration within microchannels in combined Poiseuille and electrokinetic flows, SHAURYA PRAKASH, VARUN LOCHAB, The Ohio State University — Recent discoveries show that dilute colloidal particle suspensions with diameters \(<1\mu m\) are assembled to distinct colloidal bands within microchannels (100 – 300µm wide x 34µm deep x 4cm long). Band formation requires opposing Poiseuille and electrokinetic flows. Band formation also requires a minimum applied potential threshold at a given shear rate. Band formation is a function of particle size and volume fraction, particle and channel wall zeta potential, electrolyte concentration, and the minimum electric field thresholds change non-monotonically for particle mixtures. Here, we discuss the broad parameter ranges to elucidate robustness of particle migration to and away from walls, formation of particle bands, and influence of fluid properties on particle migration and band formation. Interestingly, in co-flow with Poiseuille and electrokinetic flows, particles migrate away from the microchannel walls with aggregation near the bulk of the microchannel. Colloidal particle migration away from or towards the microchannel walls is likely due to the particle slip velocity with respect to the fluid. The particle migration is attributed to an electrohydrodynamic lift-like force, analogous to the inertial lift forces in sedimentation flows.

8:26AM H36.00003 Particle motion nearby rough surfaces, CHRISTINA KURZTHALER, AMIR PAHLAVAN, LAILAI ZHU\(^1\), HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University, NJ 08544, USA — Hydrodynamic interactions of particles with surrounding surfaces play a pivotal role in a variety of biological systems and microfluidic applications. Here, we study the motion of an externally driven particle through a viscous fluid along different boundaries that are characterized by periodic and rough surface shapes. We derive analytical expressions for the translational velocities of the particle using the Lorentz reciprocal theorem and find that the particle follows the surface shape, as anticipated by the time-reversal symmetry of Stokes flow. Moreover, our theoretical framework permits the statistical analysis of random rough surfaces, which do not affect particle motion on average but manifest themselves in the dispersion of particle trajectories. Our results are supported by numerical simulations using a boundary integral method and experimental observations. Overall, they should lay the foundation for our future understanding of microswimmer motion nearby random, heterogeneous boundaries.

1Linn Flow Centre and Swedish e-Science Research Centre (SeRC), KTH Mechanics, Stockholm, SE-10044, Sweden

8:39AM H36.00004 Hydrodynamic interactions between permeable particles, RODRIGO REBOUCAS, MICHAEL LOEWENBERG, Yale University — An analysis is presented for pairwise hydrodynamic interactions between permeable spherical particles in the limit of weak permeability, \(K = k/a^2 \ll 1\), where \(k\) is the permeability, and \(a\) is the reduced radius of the particles. Except for the near-contact motion of the particles, hydrodynamic interactions can be approximated by smooth spheres, the permeability having a weak perturbative effect under these conditions. However, non-zero particle permeability qualitatively affects the near-contact asymmetric motion, eliminating the classical lubrication singularity for smooth spherical particles. An integrodifferential lubrication equation incorporating Darcy’s law for flow inside the particles describes the near-contact motion of permeable spheres under weak permeability conditions. Collision efficiencies and hydrodynamic diffusivities are calculated and it is shown that permeable spherical particles are hydrodynamically equivalent to rough spheres with roughness \(\delta/a \approx K^{2/5}\).

8:52AM H36.00005 Spreading of diffusiophoretic colloids under transient solute gradients: Super-diffusion, trapping and shuttling\(^1\), HENRY CHU, Department of Chemical Engineering and Center for Complex Fluids Engineering, Carnegie Mellon University, Pittsburgh, PA 15213, United States, STEPHEN GAROFF, Department of Physics and Center for Complex Fluids Engineering, Carnegie Mellon University, Pittsburgh, PA 15213, United States — Diffusiophoresis (DP) refers to the deterministic drift of one species induced by a concentration gradient of another species. Recent microfluidic experiments have focused on DP of micron-scale colloids in gradients of small ionic solutes. A solute concentration gradient results in a DP colloid velocity \(\mathbf{u}_{\text{DP}} = M/\nabla \log S\), where \(M\) and \(S\) are the DP mobility and solute concentration, respectively. The mobility \(M\) can be positive or negative, corresponding to DP driving colloids up (solute-attracting) or down (solute-repelling) the solute gradient, respectively. Here, we calculate the advective-diffusive spreading of DP particles under transient solute gradients, highlighting novel transport phenomena for microscale sorting, deposition, and delivery of colloids. We show that evolution of an initial point source of colloids depends critically on the ratio of the DP mobility to solute diffusivity, with behavior ranging from spatial trapping for “solute-attracting” colloids, to long-time super-diffusion for “solute-repelling” colloids. Finally, a solute undergoing advective translation is shown to rapidly shuttle the colloids.

\(^1\)This work was funded by Procter and Gamble.

9:05AM H36.00006 Inertial bifurcation of the equilibrium position of a neutrally-buoyant circular cylinder in shear flow between parallel walls, ANDREW FOX, JAMES SCHNEIDER, ADITYA KAHR, Department of Chemical Engineering and Center for Complex Fluids Engineering, Carnegie Mellon University, Pittsburgh, PA 15213, United States — The dynamics of a neutrally-buoyant, rigid circular cylinder in shear flow between planar, parallel walls are quantified via various particle Reynolds numbers \(Re_p\) and confinement ratios \(\kappa\) via lattice Boltzmann simulations. An inertial lift force acting transverse to the ambient shear flow has a single zero crossing at the center of the channel below a critical \(Re_p\), corresponding to a single stable transverse equilibrium position. Above the critical \(Re_p\), the equilibrium position bifurcates, with an unstable equilibrium position at the center and two additional stable equilibria equidistant off-center. Trajectories of a force- and torque-free particle confirm the equilibrium position bifurcation, showing the cylinder reaches the center equilibrium position below the critical \(Re_p\) and the off-center equilibria above; the stable equilibrium position is independent of the initial cylinder position, with the lone exception of the aforementioned unstable equilibrium. The critical \(Re_p\) dependent on the confinement ratio, and thus particle size, and occurs below the transition to unsteady flow.

9:18AM H36.00007 inFocus: Fast Inertial Lift Velocity Calculation In Arbitrary Geometry\(^1\), SAMUEL CHRISTENSEN, grad student at UCLA, RAYMOND CHU, undergrad at UCLA, CHRIS ANDERSON, MARCUS ROPER, Professor at UCLA — Inertial microfluidic devices use inertial lift forces to arrange particles into required positions and formations. However, fully realized three dimensional simulations of inertial focusing require long computation times, making predictive device design very difficult. Here we use a combination of asymptotic methods and computational fluid dynamics to develop a fast, open-source fluid flow solver in Matlab that calculates particle dynamics in channels with arbitrary geometry. We use the new solver to dissect the contributions of shear and wall curvature to particle focusing with the goal of developing mechanistic understanding of how, where, and with what speed particles focus under inertial forces.

\(^1\)NSF
9:31AM H36.00008 High Frequency Inertial Particle Focusing. GIRDAR VISHWANATHAN, DIANZHOU WANG, GABRIEL JUAREZ, University of Illinois Urbana-Champaign — Inertial Focusing in micro-channels is a simple and reliable means of sorting, separating and controlling particle position, usually accomplished by producing steady flow in a long micro-channel. Recently, oscillatory flows have been shown to enable focusing of sub-micron particles, in much shorter channel lengths and at decreased pressure gradients even for frequencies < 20 Hz. Considering the substantial improvement of focusing efficiency even at relatively low oscillation frequencies, we present our experimental observations on the focusing of particles in the high frequency (20 – 1000 Hz) range. The role of the channel Womersley number on the focusing performance is critically examined.

9:44AM H36.00009 Solute-Driven Colloidal Particle Manipulation in Continuous Flows Past Microstructured channels. GUIDO BOLOGNESI, GORAN T. VLADISAVLJEVIC, Dept. of Chemical Engineering, Loughborough University, UK, FRANCOIS NADAL, Dept. of Mechanical Engineering, Loughborough University, UK, CECILE COTTIN-BIZONNE, CHRISTOPHE PIRAT, Universite Claude Bernard Lyon 1 - CNRS, France, NAVAL SINGH, Dept. of Chemical Engineering, Loughborough University, UK — Recent advancements in the chemical and biological analysis have led to the introduction of colloidal particle manipulation capabilities into microfluidic devices. Various active techniques have been employed for particle operations. The ability to manipulate particles by diffusion, a phoretic phenomenon leading to particle motion along a solute concentration gradient without the use of an external field, has gained an increasing attention. The aim of this study is to explore diffusiophoresis to enable particle filtration, trapping, and accumulation within a microfluidic environment under continuous flow settings. A microchannel, made of an optical adhesive and fitted with a micro-structured wall, was fabricated by photo-/soft-lithography. The charged fluorescent colloidal particles were accumulated within the channel microstructures by pumping electrolyte solutions into the device junction to generate salt concentration gradients. The spatial distribution of particles was characterized via confocal microscopy. This novel approach of particle handling in lab-on-a-chip device by solute driven transport can unlock potential applications in point of care industry, drug delivery and biosensing.

1 EPSCRC

9:57AM H36.00010 Numerical Study of a Particle migration in a liquid-liquid stratified flow in a microchannel. T. KRISHNAVENI, T. RENGANATHAN, S. PUSHPAVANAM, Indian Institute of Technology, Madras — Inertial focusing is a passive separation technique in an axial flow where particles migrate laterally to equilibrium positions in the presence of finite inertia. These equilibrium positions mainly depend on the two counteracting forces namely the wall lift force and the shear gradient force. The equilibrium positions can be altered by changing the velocity profile of the fluid. In this work, we model the inertial focusing of a particle in the laminar, liquid-liquid stratified flow in a microchannel. This is used for the separation and recovery of cells from one fluid to the other without using any membrane. The particle is sent through one fluid and due to the shear gradient force and interfacial force, the particle may focus into the other fluid or stick at the interface. A flow between infinite parallel plates is considered to study the particle migration. An immersed boundary method coupled with the level set method is used to study the hydrodynamics and the particle dynamics. It is observed that the particle focus in the low viscous fluid beyond a critical flowrate ratio of two liquids. The effect of other parameters like phase holdup, Reynolds number, Weber number, particle size and wettability is analyzed on the particle migration.

10:10AM H36.00011 Spontaneous Lift of Particles Through Inhomogeneous Slip Surfaces in Lubricated Contacts. AIDAN RINEHART, UGIS LACIS, Royal Institute of Technology (KTH), THOMAS SALEZ, Laboratoire Ondes et Matiere d’Aquitaine (LOMA), SHERVIN BAGHERI, Royal Institute of Technology (KTH) — We reveal how inhomogeneous surfaces can accomplish wear reduction and increased mobility for particles traveling near walls. We consider a model problem of a cylinder near a wall where the surface (cylinder or wall) has non-homogeneous slip properties. We demonstrate that through variations in surface slip length the lubrication pressure symmetry is broken. Using lubrication theory we provide the analytical solutions to the hydrodynamic force and torques acting on the cylinder. Using numerical simulations, we also report various particle trajectories arising from inhomogeneous slip surfaces, including migration, agglomeration, and self-propulsion. We find a linear scaling between wall normal migration and slip length, $\Delta \sim l$, for wall parallel motion over a slip to no-slip wall transition. These findings are relevant especially for micro-fluidic and biological systems where particles typically reside in close proximity to walls.

Monday, November 25, 2019 8:00AM - 10:36AM – Session H37 Particle Laden Flows: Non-Spherical Particles 619 - S. Balachandar, University of Florida

8:00AM H37.00001 Comprehensive modeling of hydrodynamic forces and torques on non-spherical particles using the PIEP model. W. C. MOORE, S. BALACHANDAR, YUNCHAO YANG, University of Florida — The first aim of this work is to present a general formulation for the force and torque experienced by an isolated non-spherical particle. Then, a working model is introduced for a spheroid subject to uniform flow. To construct this model, particle resolved direct numerical simulations (PRDNS) of flow around a spheroid are performed for sub-critical Reynolds numbers. These PRDNS consider various angles of attack, rotational Reynolds numbers, and axes of rotation. The pairwise interaction extended point-particle (PIEP) model is then utilized to model a spheroid as three spheres, and extensions to other geometries, such as cubes as superposition of nine spheres, are being considered. Lastly, this model is used to predict the dynamics of a settling spheroid. The model’s results are compared to PRDNS results.

1 This material is based upon work supported by the National Science Foundation Graduate Research Fellowship Program under Grant No. DGE-1315138 and DGE-1842473, by the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, under Contract No. DE-NA0002378, and by the Office of Naval Research (ONR) as part of the Multidisciplinary University Research Initiatives (MURI) Program, under grant number N00014-16-1-2617. The views and conclusions contained herein are those of the authors only and should not be interpreted as representing those of ONR, the U.S. Navy or the U.S. Government.
8:13AM H37.00002 Rotation rate of fibers in turbulence. CAUTIER VERHILLE, IRPHE - CNRS, ANKUR BORDOLOI, THERESA OEHMKE, Department of Civil and Environmental Engineering, UC Berkeley, IRPHE TEAM, UC BERKELEY TEAM — Since the last decade, more and more studies are devoted to the dynamics of anisotropic particles in turbulence. It has been shown theoretically and numerically that fibers smaller than the Kolmogorov length tend to align preferentially with the vorticity. More recently, Pujara et al. shows that longer fibers tend to align with the most extensional direction of the coarse grained velocity gradient. This difference of preferential alignment should have major impact on the rotational dynamics of particles: small fibers are expected to spin whereas long fibers are expected to tumble. We present here an experimental investigation on the global rotational dynamics (tumbling and spinning) of fibers smaller and larger than the Kolmogorov length. In this talk we will quantify the spinning and tumbling of fibers smaller and longer than the Kolmogorov length.

1 This work was carried out in the framework of FlexFT Project (ANR-17-CE30-0005-01) funded by the French National Research Agency (ANR).

8:26AM H37.00003 Dynamics and wakes of freely settling and rising cubes. ANTHONY WACHS, University of British Columbia, ARMAN SEYED-AMMADI, Department of Chemical & Biological Engineering, University of British Columbia — In this investigation, we present numerical simulations of freely settling and rising cubes in a quiescent Newtonian fluid for various values of the Galileo number $70 \leq Ga \leq 250$ and of the solid-to-fluid density ratio $0.2 \leq \rho \leq 7$. Ultimately, we obtain a comprehensive two-parameter flow map for a freely moving cube and characterize prominent features of each regime of motion such as trajectories and wake structure. Unlike the case of a sphere, helical motion is observed for all density ratios, marking it as a characteristic type of motion for a cube. Furthermore, we present an in-depth force analysis relevant to the induced lateral motions, and we show that there is a significant jump in the drag coefficient coincident with the onset of the helical regime where large-amplitude lateral displacements appear. The enhancement of the drag coefficient is explained to be a combined effect of the vortex-induced drag and the orientation angle of the cube.

1 Natural Sciences and Engineering Research Council of Canada.

8:39AM H37.00004 Shape Dependence of Settling Velocities for Particles in Wavy Flows. LAURA CLARK, Stanford University, MICHELLE DIBENEDETTO, Woods Hole Oceanographic Institute, NICHOLAS OUELLETTE, JEFFREY KOSEFF, Stanford University — We investigated the effect of particle shape and oscillatory flow on the settling velocities of negatively buoyant particles with intermediate particle Reynolds numbers. In a laboratory wave tank, we tested a range of particle shapes, including discs, rods, and spheres, and wave characteristics, including shallow, deep, and intermediate surface gravity waves. By simultaneously measuring the velocities of the non-spherical particles and the fluid flow field, we extracted the instantaneous particle slip velocities, the statistics of how the particles preferentially sample the wave field, and their net settling velocities. We find that all of these quantities display a significant shape dependence.

1 This work was supported by the US National Science Foundation under grant no. CBET-1706586.

8:52AM H37.00005 On the free-fall dynamics of highly inertial ellipsoids at $Re_p$ of $O(1)$. JOHANNES MILAN GÜTTLER, GHOLAMHOSSEIN BAGHERI, Max Planck Institute for Dynamics and Self-Organization — Much is still unknown about the dynamics of inertial ellipsoids as inertial non-spherical particles in turbulence lag behind the flow to adjust their orientation. In particular, it is not quantitatively understood how particles of different shapes orient themselves in turbulent flows and how fast their orientation responds to flow fluctuations. Experiments on single particles in a quiescent medium are the first step to characterize this. We present our work on freely falling ellipsoidal particles in the ‘intermediate’ regime of particle Reynolds number 1-10 using shadowgraphy in an air-filled column setup. The particles are well-defined and have volumes equivalent to a diameter 140 μm sphere, which became possible thanks to utilizing 2-Photon-Polymerization. The density ratio between the particles and the medium (air) is approx. 1000. The experiments were performed using three high-speed cameras to capture both the transient and the terminal state. This allows us to look into rotation/tumbling rates, terminal velocities and the transient dynamics: do stable fixed points in the orientation or oscillatory motions exist and is there a stable orientation that the particle will take? For particles with unstable transient orientation, what is the steady-state orientation?

1 Marie Skłodowska-Curie Actions, Grant Agreement No. 675675.

9:05AM H37.00006 Taylor-Aris Dispersion of Elongated Rods. AJAY HARISHANKAR KUMAR, THOMAS POWERS, DANIEL HARRIS, Brown University — Particles transported in fluid flows, such as cells or nanorods, are rarely spherical in nature. In this study, we numerically and theoretically investigate the dispersion of an initial concentration of elongated rods in 2D pressure-driven shear flow. The rods translate due to diffusion and advection, and rotate due to rotational diffusion as well as their classical Jeffery’s orbit in shear flow. When rotational diffusion dominates, we approach the classical Taylor Dispersion result for the longitudinal spreading rate by using an orientationally averaged translational diffusivity for the rods. However, in the high shear limit, the rods tend to align with the flow and ultimately spread faster as a direct consequence of their anisotropic diffusivities. The relative importance of the shear-induced orbit and rotational diffusivity can be represented by a rotational Peclet number, and allows us to bridge these two regimes.

1 We acknowledge funding received from NSF through award CMMI-1634552.

9:18AM H37.00007 Tuning the hydrodynamic interactions between non-uniform sedimenting particles. KAVINDA NISSANNA, XIAOLEI MA, JUSTIN BURTON, Emory University — Sedimentation dynamics of non-Brownian particles with non-uniform density distributions have recently been shown to exhibit hyperuniformity (Goldfriend et al., Phys. Rev. Lett., 2017). These predictions are tested here experimentally using spheroidal particles. Particles are classified by two parameters, their aspect ratio $\kappa$ and center of mass offset $\chi$. They were created from 2mm diameter spheres glued in various configurations. Spheres of different densities are used to vary $\chi$. We record the positions of particles using high-speed camera, as they sediment in a quasi-2D tank at $Re = 10^{-3}$. Using tracking algorithms, we reconstruct the sedimentation trajectories. Pair interaction between particles is characterized by particle center separation vs. vertical distance traveled. Particles with $\kappa > 1$ exhibit repulsive behavior, while particles with $\kappa < 1$ attract each other. Interaction strength is shown to be inversely proportional to $\chi$. For 3 or more simultaneously sedimenting particles, repulsive interactions lead to a more uniform distribution of particles positions, whereas attractive interactions lead to particle clustering.

1 This work was supported by the National Science Foundation DMR Grant No. 1455086.
Circulation significantly changes the flow topology into a bidimensional state so as to be responsible for the rotation rate variation.

The ratio shows a significant decrease. We propose a simple model to qualitatively understand the phenomenon. It is found that the large scale isotropic turbulence, the global rotation rate square for particles reveals a similar distribution whereas the averaged value as a function of aspect studies. Experiments as well as numerical simulations have highlighted their complex behavior, which is inherited from the non-trivial dynamics.

Flows, LEE WALSH, GREG VOTH, Wesleyan University — In a suspension of non-attractive rigid rods in a turbulent shear flow, we observe spontaneous aggregation into traveling clusters of high concentration. The fibers first accumulate in the turbulent boundary layer where they then break the symmetry of the apparatus by collecting into a few traveling clusters. The experimental apparatus is a Taylor–Couette cell whose floor and inner wall are rotating, and ceiling and outer wall are fixed. The heavy particles sediment downward and outward, so their concentration is highest near the junction of the rotating floor and the fixed outer cylindrical wall. With a multi-camera imaging system we can measure 3D fluid and fiber motion in the entire volume, and we are able to resolve individual fiber dynamics in the high-concentration regime where fiber-fiber interactions become important.

Time scale of rotation of inertial fibers in isotropic turbulence, ANKUR BORDOLOI, University of California Berkeley, GAUTIER VERHILLE, Aix Marseille Univ, CNRS, Centrale Marseille, IRPHE, EVAN VARIANO, University of California Berkeley — The past few years of research has significantly advanced our understanding of the rotational kinematics of inertial fibers in turbulence and their scaling with size. However, the question related to the time scale of rotational dispersion remains open. The time scale of rotation of inertial fibers is integral to processes such as paper-making, turbulent drag-reduction, fiber-glass blowing, and locomotion of planktonic organisms in the ocean. Based on time-resolved measurements of the orientation of rigid inertial fibers in a turbulence-tank, we compute a time scale \( \tau_1 \), fiber rotation. We show that this time-scale can be predicted by Kolmogorov’s inertial-range scaling based on fiber length \( L \) only when the diameter \( d \) is small. For fibers with large diameters, we invoke our previous theoretical model designed specifically for the variance of fiber rotation and find that it fails to predict \( \tau_1 \). We propose herein a new model that successfully predicts both these parameters for a wide range of \( L \) and \( d \).

Inertial Settling of Flexible Fiber Suspensions, MONA RAHMANI, University of British Columbia, ARASH ALIZAD BANAEI, KTH Royal Institute of Technology, MARK MARTINEZ, University of British Columbia, LIFENG JIANG, CHAO SUN,Tsinghua University, 100084 Beijing, China, ENRICO CALZAVARINI, Universite de Lille, F 59000 Lille, France — The rotational dynamics of small anisotropic material particles (e.g. fibers or disks) in turbulent flows has been the focus of a series of recent studies. Experiments as well as numerical simulations have highlighted their complex behavior, which is inherited from the non-trivial dynamics of the velocity gradient tensor along the particle trajectories. We report the investigation of orientation dynamics of neutrally buoyant anisotropic particles as they are advected in the Rayleigh-Benard convection by means of experiments and simulations. Compared with the homogeneous isotropic turbulence, the global rotation rate square for particles reveals a similar distribution whereas the averaged value as a function of aspect ratio shows a significant decrease. We propose a simple model to qualitatively understand the phenomenon. It is found that the large scale circulation significantly changes the flow topology into a bidimensional state so as to be responsible for the rotation rate variation.

Tumbling rate of anisotropic particles in turbulent convection, LINFENG JIANG, CHAO SUN, Tsinghua University, 100084 Beijing, China, ENRICO CALZAVARINI, Universite de Lille, F 59000 Lille, France — The rotational dynamics of small anisotropic material particles (e.g. fibers or disks) in turbulent flows has been the focus of a series of recent studies. Experiments as well as numerical simulations have highlighted their complex behavior, which is inherited from the non-trivial dynamics of the velocity gradient tensor along the particle trajectories. We report the investigation of orientation dynamics of neutrally buoyant anisotropic particles as they are advected in the Rayleigh-Benard convection by means of experiments and simulations. Compared with the homogeneous isotropic turbulence, the global rotation rate square for particles reveals a similar distribution whereas the averaged value as a function of aspect ratio shows a significant decrease. We propose a simple model to qualitatively understand the phenomenon. It is found that the large scale circulation significantly changes the flow topology into a bidimensional state so as to be responsible for the rotation rate variation.

A finite Re slender body theory, ANUBHAB ROY, Department of Applied Mechanics, Indian Institute of Technology Madras, Chennai 600036, India, DONALD L. KOCH, Smith School of Chemical and Biomolecular Engineering, Cornell University, Ithaca, NY 14853, USA — The effects of fluid inertia on the settling motion of fibers is studied theoretically. Khayat & Cox (1989) were the first to give a theory of hydrodynamic forces and torques on a slender body when fluid inertia is non-zero. Their theory uses a matched asymptotic expansion with a viscous inner flow and Oseens approximation for the outer flow. This restricts the analysis to cases where Re defined based on fiber diameter \( \text{Re}_D \) is zero. We develop a novel finite Re slender body theory that allows the inner flow to be described by steady Navier-Stokes and thus provide better comparisons of drag and torque with realistic scenarios where the \( \text{Re}_D \neq 0 \).

Monday, November 25, 2019 8:00AM - 10:36AM – Session H38 Porous Media Flow General I 620 - Chris MacMinn, University of Oxford

Soft porous lubrication with oriented fiber array, ZENGHAO ZHU, QIANHONG WU, Villanova University — The friction mitigating property of oriented fibrous materials is widely observed in nature. In this paper, we propose a novel theoretical model to predict the lubrication performance due to the gliding motion of a flat surface over a soft porous layer made of oriented fibers. The lubrication theory for highly compressible porous media is used to predict the lifting force from the fluid phase. The theory for large deformation of flexible beam is used to predict the lifting force from the solid phase. The results are validated numerically using Abaqus. The relative contribution of the fluid lifting force to the overall lifting force, fp, is then obtained, which is used as the criteria to evaluate the lubrication performance using soft porous materials. Dimensional analysis, along with a comprehensive parametric study have been performed to reveal the dominant factors that determine the lubrication performance. The study presented herein provides valuable guidance for applying highly organized porous media to soft porous lubrication.

1National Science Foundation, Award No. 1511096.
8:13AM H38.00002 Capturing gas in soft granular media. SUNGYON LEE, Department of Mechanical Engineering, University of Minnesota, Minneapolis, Minnesota, 55455, USA, JEREMY LEE, Department of Engineering Science, University of Oxford, Parks Road, Oxford OX1 3PJ, UK, FENG XU, Department of Mechanical Engineering, University of Minnesota, Minneapolis, Minnesota, 55455, USA, CHRISTOPHER MACMINN, Department of Engineering Science, University of Oxford, Parks Road, Oxford OX1 3PJ, UK — Gas migration through a soft granular material involves a strong coupling between the motion of the gas and the deformation of the material. We study this process experimentally by injecting air into a quasi-2D packing of soft particles and measuring the morphology of the air as it rises due to buoyancy. We systematically increase the confining pre-stress in the packing by compressing it with a fluid-permeable piston, leading to a gradual transition in migration mechanism from fluidization to pathway opening to pore invasion. By connecting these mechanisms quantitively with macroscopic invasion, trapping, and venting, we show that the mixed-mode transitional regime enables a sharp increase in the amount of gas trapped within the packing, as well as much larger venting events. We report our experimental findings and present a simple mechanistic model to rationalize our observations.

1The support from EPSRC (UK) under grant EP/P020860/1 is gratefully acknowledged

8:26AM H38.00003 Regularization of singularities at corners in two-dimensional Darcy flows1, ALEXANDER BELOZEROV, NATALIA PETROVSKAYA, YULII SHIKHMURZAEV, School of Mathematics, University of Birmingham, UK — As can easily be shown, for flows past wedges protruding into a porous medium the standard Darcy model produces unphysically singular velocities, thus over-predicting the flow rate and making impossible to describe some flows, e.g. involving wetting fronts, where the pointwise distribution of velocity has to be modelled realistically. A seemingly obvious remedy of smoothing out corners does not apply as, in the real life, the radius of curvature of the tips of such corners can easily be on the pore scale, i.e. zero on the Darcy scale. A recent study (AIChE J. 63(2017)5207) introduces an approach to the problem where, following a suggestive physical analogy, the permeability even of a spatially uniform porous matrix with respect to a flow is, at every point, a function of the curvature of the flow streamline at this point, decreasing as the curvature increases. This preserves the Darcy model for unidirectional flows, where it has been well-tested, and regularizes 2D flows, where the flow field and the distribution of permeability now become intertwined and have to be found simultaneously. The new class of models brings in a fundamentally new class of numerical problems. In the present work, we develop a method of handling such problems and highlight some outstanding issues.

8:39AM H38.00004 A comprehensive experimental and analytical study of fluid flow in a thin porous layer under indentation.1, QIUYUN WANG, ZENGHAO ZHU, QIANHONG WU, Villanova University — In this paper, we report a novel experimental and theoretical approach to examine fluid flow in a thin porous layer during an indentation process. Experimentally, a custom-designed indenter with the precisely controlled nano-positioning system is developed where the local compression of the porous layer is captured by a high-speed camera. The indenter is fully instrumented with a laser displacement sensor to measure the indentation velocity and pressure transducers to measure the pore pressure distribution. Theoretically, a consolidation theory is developed where the local relative velocity between the fluid and solid phase is considered, and the local compression-dependent permeability dependent on the indentation velocity. Excellent agreement between the experimental results and the theoretical predictions is observed under different running conditions, verifying the validity of the theory. The study precisely captures the detailed non-uniform compression of a thin porous layer under indentation. It provides a conclusive theoretical framework in the study of fluid flow in a soft porous media, laying the foundation for the study of soft porous matter in response to indentation.

1NSF CBET fluid dynamics program, #1511096

8:52AM H38.00005 The hydraulic conductivity of shaped fractures with permeable walls1, DAIHUI LU, IVAN C. CHRISTOV, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana, 47907 — We derive the hydraulic conductivity $K$, i.e., the proportionally constant between the width-averaged velocity field and the pressure gradient in Darcy’s law, for shaped fractures with permeable walls. As a model, we study a tapered Hele-Shaw cell, with a width gradient $dh/dx = \alpha$ in the flow direction, and porous boundaries. The permeable walls are treated using the Beavers–Joseph slip boundary condition. Using lubrication theory, we obtain $K$, accounting for geometric non-uniformity and leakage into the bounding surfaces. The approach is perturbative, giving both the leading-order term (independent of the Reynolds number $Re$) and the first correction in $Re$. Thus, our theory gives $K$ in terms of hydraulic parameters such as $Re$, geometric parameters such as the fracture’s width $h(x)$ and $\alpha$, and the dimensionless slip coefficient $\phi$ at the porous walls. Previous research has not addressed the joint dependence on $Re$ and $\alpha$. Specifically, our calculations show that, quantitatively, $Re$ has a comparable effect to $\phi$ on the value of $K$, for $\alpha \neq 0$. Finally, we use the open-source computational fluid dynamics software, OpenFOAM, to perform 3D direct numerical simulations to benchmark and verify our mathematical predictions.

1This work was supported by ACS PRF grant 57371-DN19.

9:05AM H38.00006 Stochastic Modelling of Sieving in Membrane Filters with Complex Pore Morphology1, BINAN GU, New Jersey Institute of Technology, PEJMAN SANAEI, New York Institute of Technology, LOU KONDIC, LINDA CUMMINGS, New Jersey Institute of Technology — Membrane filters have been widely used in industrial applications to remove contaminants and undesired impurities from the solvent. During the filtration process the membrane internal void area becomes fouled with impurities and as a consequence the filter performance deteriorates. In addition to the internal morphology of membrane filters, fouling mechanisms contribute to the complexity of the filtration process, in deterministic or stochastic ways. So far various models have been proposed to describe the membrane structure and stochasticity of particles flow individually but very few focus on both together. In this work, we present a model, in which a membrane consists of a series of bifurcating pores, which decrease in size as the membrane is traversed and particles are removed from the feed by adsorption within pores and sieving simultaneously. We derive a probabilistic formulation of the sieving process using a continuous-time Markov chain. Lastly, we discuss how filtration efficiency depends on the characteristics of the branching structure and show the coupling between the two fouling mechanisms.

1NSF-1615719
9:18AM H38.00007 Upscaled models for heterogeneous reactions in porous media¹, MATTEO ICARDI, FEDERICO MUNICCHÌ, University of Nottingham — Although the basic understanding of the macroscopic limit of linear advection diffusion reaction equations is well understood since the early developments of porous media theory, its extension to complex flow regimes is still currently an open question, even in presence of well-separated spatial scales. This is due to the presence of non-trivial microscopic equilibrium configurations (compared to the trivial constant solution obtained by standard periodic homogenisation), or of dynamic equilibrium configurations. For example, when dealing with fast surface reactions, large microscopic gradients can develop locally. Similarly, a conservative solute in the vicinity of a concentration source (injection) undergoes a dynamic evolution of the local microscopic configuration in time and space before reaching the asymptotic self-similar profile. These are only two examples when the classical upscaling approaches fail, and effective macroscopic equations are often found either empirically or by resorting to generic random walk models. In this talk, we present some (old and) new theoretical frameworks to overcome these limitations, by computing local spectral properties (eigenvalues and eigenfunctions) of the underlying transport operators.

¹This work has been funded by the European Unions Horizon 2020 research and innovation programme, grant agreement number 764531, "SECURE Subsurface Evaluation of Carbon capture and storage and Unconventional risks".

9:31AM H38.00008 A Theoretical Model of Flow in a Pleated Filter Membrane Cartridge¹, DANIEL FONG, US Merchant Marine Academy, PEJMAN SANAEI, New York Institute of Technology — A 3D model is developed for fluid flow in a cylindrical pleated membrane filter cartridge consisting of a central hollow region surrounded by annular pleated membrane within an external housing. In the region outside of the pleated filter and the hollow region, flow is described using Stokes equation, while in the pleated membrane region Darcy’s law is used to model flow through the porous pleated membrane. The governing equations are reduced by exploiting the small aspect ratios of both the membrane and housing. This yields a coupled simplified system of equations for the pressures and velocities in these three regions which is solved numerically. The model is used to investigate the effect of changing various pleated membrane cartridge geometrical parameters on the filter efficiency.

¹P.S. gratefully acknowledge support from NSF RTG/DMS-1646339

9:44AM H38.00009 Liquid flow through paper with the consideration of swelling and intra-fiber pores¹, WONJUNG KIM, SOOYOUNG CHANG, Sogang University — Paper is one of the most widely used porous media for absorbing liquid, and accurate control of water imbibition in paper is crucial in developing paper-based microfluidic devices. Washburn equation is usually used to describe the dynamics of the liquid flow through the cellulose matrix of paper. However, it is well known that this equation has limitations in predicting water flow in paper. We report that swelling and intra-fiber pores that have not been considered in developing Washburn equation are mainly responsible for the limited accuracy when predicting imbibition length of water. We demonstrate that cellulose fibers have significant internal voids that absorb water. In addition, we quantify water induced swelling that leads to the expansion of inter-fiber space. We develop a hydrodynamic model of water imbibition with the consideration of intra-fiber voids and cellulose fiber swelling that well explain experimental observations. Our study provides a new insight into not only porous media flow with intra-void structure and swelling effects, but also a theoretical background to design μPADs.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (grant no 2017R1A1A01073599)

9:57AM H38.00010 Pressure-driven flow across a hyperelastic porous membrane,¹ RYUNGEUN SONG, Sungkyunkwan University, HOWARD STONE, Princeton University, KAARE JENSEN, Technical University of Denmark, JINKEE LEE, Sungkyunkwan University — We report an experimental investigation of pressure-driven flow of a viscous liquid across thin polydimethylsiloxane (PDMS) membranes. Our experiments revealed a nonlinear relation between the flow rate Q and the applied pressure drop ∆p, in apparent disagreement with Darcy’s law, which dictates a linear relationship between flow rate, or average velocity, and pressure drop. These observations suggest that the effective permeability of the membrane decreases with pressure due to deformation of the nanochannels in the PDMS polymeric network. We propose a model that incorporates the effects of pressure-induced deformation of the hyperelastic porous membrane at three distinct scales: the membrane surface area, which increases with pressure, the membrane thickness, which decreases with pressure, and the structure of the porous material, which is deformed at the nano-scale. With this model, we are able to rationalize the deviation between Darcy’s law and the data. Our result represents a novel case in which macroscopic deformations can impact the microstructure and transport properties of soft materials. ¹JL thanks to the support by NRF (2017R1A2B2006964) and KEITI (2019002790003). HAS thanks the NSF for support via grant CMMI-1661672 and DMR-1420541. KJH was supported by a research grant (13166) from VILLUM FONDEN.

10:01AM H38.00011 Nonlinear Fluid-Structure Coupling in Flow through a Deformable Porous Medium,¹ TYLER LUTZ, LARRY WILEN, JOHN WETTLAUFER, Yale University — Fluid flow through a deformable, porous matrix of solid particles is generally seen to exhibit non-Darcy behavior: viscous drag forces exerted on the solid constituents by the fluid may be communicated throughout the solid matrix, leading to a spatially inhomogeneous permeability and ultimately a nonlinear relationship between bulk pressure drop and volume flux. We present results of an experiment in three dimensions designed to provide a closure condition on the mathematical theory of large-deformation flow through a deformable porous medium. Our specific realization of uniaxial flow through a cylindrical foam is robust enough to capture the nonlinear coupling between the solid and fluid components, while the geometry is sufficiently simple to enable detailed comparisons to theoretical expectations. We focus in particular on precision measurements of the pore pressure gradient as a function of the driving pressure head; we use these measurements in combination with direct measurements of the foam deformation to determine the relationship between porosity and permeability, a crucial theoretical input parameter. This closure condition allows us to make explicit, quantitative comparisons to theoretical predictions.

10:23AM H38.00012 Random Resistor Networks and Porous Media, AHMAD ZAREEI, SHIMA PARS, ARIEL AMIR, Harvard University — Pore-level flow distribution in a disordered porous medium is determined through its heterogeneous micro-structure. Dissolution of solid matrix or solute retention during flow of a solution dynamically changes the micro-structure, affects the velocity distribution of pores in unpredictable ways, and further changes the bulk behavior. In order to understand the dynamics of local structural changes during polymer solution flow and its effects, we investigate random resistor networks as a model of disordered porous medium. The network of pores is modeled using connected network of pipes with a random distribution of diameters, and then the critical behavior and averaging behavior of such networks are developed. We further study the dynamics of structural changes during polymer retention process in the random resistor network, and show how this model is able to predict local process of polymer retention, micro-structural changes, flow velocity distribution of pores, and bulk properties consistent with experiments.
8:00AM H39.00001 Turbulent mixing in oceanic flows: challenges and insights for improved prediction. KARAN VENAYAGAMOORTHY, Colorado State University — The understanding and quantitative prediction of diapycnal (irreversible) mixing of density and momentum in the oceans remains an important ongoing challenge. From a practical perspective, there is a critical need to obtain accurate prediction of turbulent heat, mass and momentum fluxes using indirect measurements in the field. Indirect methods for estimating mixing rates typically rely on the inference of three key quantities namely: (i) the rate of dissipation of turbulent kinetic energy; (ii) the mixing efficiency, which is a measure of the amount of turbulent kinetic energy that is irreversibly converted into background potential energy; and (iii) the background density stratification, respectively. In this talk, an overview on how these quantities are typically inferred and/or parameterized will be presented. Some important challenges, ambiguities and new insights will also be presented with an eye toward improved prediction of ocean mixing.

1 Funded by ONR and NSF

8:13AM H39.00002 Predicting regions of ocean vertical transport via surface coherent structures. MICHAEL ALLSHOUSE, H M ARAVIND, Northeastern University — Vertical transport in the upper ocean impacts the surface mixing, advection of nutrients, and the ocean energy budget. Observing regions of significant vertical transport is difficult because vertical velocities in the ocean are often orders of magnitude smaller than horizontal velocities. Some tools for predicting where large vertical velocities will occur include HF radar, satellite altimetry, and modeled horizontal velocity fields, which all provide ocean surface velocities. While Eulerian analysis of these fields can yield some information, Lagrangian coherent structures are more robust to noisy observational data and model parameter uncertainty. We correlate surface coherent structures to vertical transport below the surface to evaluate their capacity to predict regions of strong vertical transport. In particular, we compute the finite-time Lyapunov exponent field from the surface velocity and compare this with the corresponding local vertical subduction. This correlation is tested on a high-fidelity simulation of a sheared subsmesoscale flow and an operational ocean forecast. The identification of coherent structures provides a target zone for anticipated vertical transport that could be observed via Lagrangian floats.

1 Support provided by ONR N00014-18-1-2790

8:26AM H39.00003 Oceanic Sub-mesoscale Wave-Vortical Interactions and Their Effect on Scalar Transport. GERARDO HERNANDEZ-DUENAS, National Autonomous University of Mexico, PASCALE LEULONG, NorthWest Research Associates, LESLIE SMITH, Department of Mathematics, University of Wisconsin - Madison — The mechanisms driving lateral dispersion in the ocean on scales of 100 m − 10 km remain, by and large, not well identified. Dominant motions in this regime, known as the subsmesoscale, are internal waves and vortical motions. These two components have similar spatial scales but evolve on different temporal scales. Small-scale vortical mode are susceptible to instabilities and may not be as long-lived as their larger-scale geostrophic counterparts. While vortices are more efficient at dispersing a passive tracer than waves, the role of the latter remains less well understood. In this talk, we will present simulations using a set of intermediate models to identify the role of various non-linear interactions between vortical and wave motions. These intermediate models range from the quasi-geostrophic model which only includes PV/PV/PV nonlinearities and GGG model with only wave/wave/wave nonlinearities to the full Boussinesq model which retains all. Statistics such as energy transfer spectra and diffusivity will be shown to identify the effect of different non-linear interactions on scalar transport.

1 UNAM-DGAPA-PAPIIT IN113019

8:39AM H39.00004 The Energetics of Seamount Wakes. BRAD PERFECT, NIRNIMESH KUMAR, JAMES RILEY, University of Washington — This work revisits the longstanding hypothesis that seamounts are the "stirring rods" of the ocean. Specifically, it has been proposed that the eddy motions generated by the interaction of underwater mountains with ocean currents might play a significant role in the broader field of ocean mixing. We report on the results of a series of numerical simulations for an idealized seamount in rotating, stratified flow at a range of Froude and Rossby numbers (0.014 < Fr < 0.14 and 0.053 < Ro < 0.21). This parameter space is consistent with the strongly stratified, strongly rotating flow experienced by a large seamount. In each simulation, the seamount generates a distinctive wake and produces internal wave radiation. We explore the energetics of the wake and waves separately, and find that the wake is predominantly controlled by the Burger number, while the internal wave radiation has a more complex dependence that can be predicted by modifying pre-existing linear wave models. Ultimately, our results suggest that for low-Froude number flows, the wake of the seamount extracts more energy from the mean flow than the internal wave flux, supporting the stirring rod view of seamount energetics.

8:52AM H39.00005 Observations of nonlinear internal wave evolution and mixing from the shelf to the surf zone. KRISTEN DAVIS, GREGORY SINNETT, EMMA REID, University of California, Irvine, Civil & Env. Engineering — Internal waves strongly influence the physical and chemical environment of coastal ecosystems worldwide. We report novel observations from a distributed temperature sensing (DTS) system that tracked the transformation of internal waves from the shelf break to the surf zone over a shelf-slope region of a coral atoll in the South China Sea. The spatially-continuous view of the near-bottom temperature field provided by the DTS offers a perspective of physical processes previously available only in laboratory settings or numerical models. Additionally, we report observations of turbulent dissipation during the passage of a shoaling internal wave train and examine the implications for irreversible mixing of subthermocline water into the nearshore region and onto a shallow coral reef. We find that during summer, internal waves shoaling on the shallow atoll regularly transport cold, nutrient-rich water shoreward, altering near-surface water properties on the fore reef. This water is transported shoreward of the reef crest by tides, breaking surface waves and wind-driven waves, where it significantly alters the water temperature and nutrient concentrations on the reef.
Towards an accurate parameterization of mixed layer deepening during ocean convection. TAIMOOR SOHAIL, The Australian National University, BISHAKHDATTA GAYEN, The University of Melbourne, ANDREW HOGG, The Australian National University — We investigate the growth and equilibrium of the oceanic mixed layer during a deep convective event using a Direct Numerical Simulation (DNS) and a large-scale ocean model. The DNS resolves all scales of flow, providing the opportunity to quantify vertical buoyancy flux and mixing in a deep convective event. The inferred vertical diffusivity and mixed layer depth (MLD) are simulated over a range of background stratifications and surface buoyancy forcings in the DNS model. A scaling theory for convective mixing is proposed which robustly predicts the rate of MLD growth and vertical diffusivity during deep convection. We directly compare the MLD predicted from the scaling theory with existing vertical parameterizations in a large-scale ocean model. Specifically, we run analogous deep convection experiments with the Modular Ocean Model (MOM) 6 running three different vertical parameterizations for the ocean boundary layer: CVMix, KFP and ePBL. We quantify the difference between the MLD derived from these parameterization schemes and the predictions from the theoretical scaling, thereby quantifying the accuracy of existing parameterization schemes.

Impacts of inertial/symmetric instabilities on ocean fronts1. NICO-LAS GRISOUARD, University of Toronto — Oceanic submesoscale density fronts are structures in geostrophic and hydrostatic balance. They tend to be small (100 m to 10 km wide at mid-latitudes) and ubiquitous features of the near-surface of the oceans. They tend to be unstable, which may be key to understanding the kinetic energy budget of the ocean, as well as their effects on gas and nutrient exchanges between the surface and the abyss. In this presentation, we focus on the inertial or symmetric instability. We present a series of numerical experiments to investigate energetic impacts of these instabilities on fronts. Our set of experiments covers the submesoscale portion of a three-dimensional parameter space consisting of the Richardson and Rossby numbers, and a measure of stratification or latitude. We first argue that contrary to parameterization prototypes that are currently being developed, drainage of available potential energy from the geostrophic flow can be a leading-order source of their growth. We also argue that a front is relatively robust when experiencing this instability, and provide hints as to its contribution to the shape of fronts. We also caution modellers about a possibly large impact of the choices of the dissipation operator on the dynamics of the instability.

Equilibration of symmetric instability and inertial oscillations at an idealised submesoscale front. AARON WIENKERS, DAMTP, University of Cambridge, LEIF THOMAS, Stanford University, JOHN TAYLOR, DAMTP, University of Cambridge — Submesoscale fronts with large lateral buoyancy gradients and O(1) Rossby numbers are common in the upper ocean. These fronts are associated with large vertical transport and are hotspots for biological activity. Submesoscale fronts are susceptible to symmetric instability (SI) — a form of stratified inertial instability which can occur when the potential vorticity is of the opposite sign to the Coriolis parameter. Growing SI modes eventually break down through a secondary shear instability, leading to 3D turbulence and vertically mixing the geostrophic momentum. Once out of thermal wind balance, the front undergoes inertial oscillations which can drive further small-scale turbulence. Here, we consider the idealised problem of a balanced front with uniform horizontal buoyancy gradient and bounded by flat no-stress horizontal surfaces. We study the equilibration of this unstable front using a linear stability analysis and 3D numerical simulations. We find drastically different behavior emerging at late times. While weak fronts develop frontlets and excite subinertial oscillations, stronger fronts produce bore-like gravity currents. We describe the details of these energy pathways as the front evolves towards the final adjusted state in terms of the dimensionless front strength.

Global Distribution of the Oceanic Bottom Mixed Layer Thickness1, PENG-QI HUANG, XIAN-RONG CEN, YUAN-ZHENG LU, SHUHANG-XI GUO, SHENG-QI ZHOU, State Key Laboratory of Tropical Oceanography, South China Sea Institute of Oceanology, Chinese Academy of Sciences, Guangzhou, China — With the increasing number of available potential energy from the geostrophic flow can be a leading-order source of their growth. We also argue that a front is relatively robust when experiencing this instability, and provide hints as to its contribution to the shape of fronts. We also caution modellers about a possibly large impact of the choices of the dissipation operator on the dynamics of the instability.

Setting of randomly formed marine aggregates. EUNJI YOO, SHILPA KHATRI, FRANCOIS BLANCHETTE, Applied Mathematics, University of California, Merced — Setting marine aggregates plays an important role in transporting carbon from the surface ocean to the deep ocean. Investigation of settling rates is critical to understand the ecological importance of these particles. We study the setting of Diffusion-Limited-Agregates as a model of marine aggregates in the ocean. The aggregates are assembled as a collection of cubic particles formed by cluster-to-cluster aggregation, resulting in fractal objects. The stresses on the surface and flow around the aggregates are computed in the limit of zero Reynolds number using a single-layer potential boundary integral method and we handle the challenges of singularities analytically. We first validate and analyze the performance of our numerical method. We then present the statistical distribution of drag and torque on aggregates of various sizes. We determine that the gyration radius is the key project (201804020056), and the Research Programs of the Chinese Academy of Sciences (XDA11030302 and ISEE2018PY05).

A Brownian dynamics model for the formation of marine aggregates. FRANCOIS BLANCHETTE, CHANGHO KIM, UC Merced — Marine aggregates, the largest of which are called marine snow, play an important role in the oceanic carbon cycle. As microorganisms form, grow, and die near the ocean surface, they tend to cluster and form aggregates. We study numerically the formation of these aggregates using a Brownian dynamics model. The mobility tensor of each aggregate particle, which significantly depends on its shape and size, is computed by a boundary integral method to solve the corresponding Stokes equations. Thus, our model provides a more accurate description of the formation mechanism of aggregates. We investigate the fractal dimension and size distribution of aggregates and also characterize the settling speed as a function of a properly defined size of an aggregate.
8:00AM H40.00001 The Age of a Wake, DAVID LEWIS, TIMOUR RADKO, None — This study attempts to quantify decay rates of stratified wakes in active oceanic environments, characterized by the presence of intermittent turbulence and double-diffusive convection. The investigation is based on a series of direct numerical simulations of wakes produced by a sphere uniformly propagating in stratified two-component fluids. We examine and compare the evolution of wakes in fluid systems that are initially quiescent, double-diffusively unstable, or contain preexisting turbulence. Model diagnostics are focused primarily on dissipation of turbulent kinetic energy ($\epsilon$) and thermal variance ($\chi$). Analysis of decay patterns of $\epsilon$ and $\chi$ indicates that microstructure generated by an object of $D = 0.6$ m in diameter moving at the speed of $U = 0.02$ m/s could be detected, using modern high resolution profiling instruments, for 0.5–0.7 h. Convective overturns are shown to be particularly effective in terms of dispersion of microscale wake signatures. Extrapolation of model results to objects of $\sim 10$ m in diameter propagating with speeds of $\sim 10$ m/s suggests that this form of detection is feasible for at least 4 h after the object’s passage through the monitored areas.

8:13AM H40.00002 Characterization of the Far-Wake of an Inclined 6:1 Prolate Spheroid, JONATHAN PECK, ETHAN LUST, U.S. Naval Academy — The study of flow around prolate spheroids has informed our understanding of fundamental hydrodynamics. For example, they have been used to study boundary layer development, turbulent flow transition, and vortex shedding. Because of the simplicity of the model and the rich hydrodynamic result, prolate spheroid data are often used to validate computational fluid dynamics codes. There are many examples from the literature featuring spheroids in various configurations: aspect ratio (L:D), Reynolds number, and inclination angle. However, there is little consideration of the far-wake, at downstream diameters greater than 6D, particularly at Reynolds numbers in excess of $4 \times 10^6$. The focus of the present study was on this region. The experiment was conducted in the large towing tank at the U.S. Naval Academy. The 6:1 prolate spheroid, measuring 54 in. in length and 9 in. in diameter, was sting-mounted at a 20° angle of inclination. An underwater, stereo particle image velocimetry system was attached to a structure fixed at a point approximately half the length of the towing tank. The spheroid was towed through the field of view in a number of configurations. The field of view was oriented laterally, capturing the strength, size, and evolution of the trailing vortex.

8:26AM H40.00003 Spreading and decay of axisymmetric drag wakes: theory, SCOTT WUNSCH, Johns Hopkins University, THEODORE D. DRIVAS, Princeton University, D. CURTIS SAUNDERS, Johns Hopkins University — The spreading and decay of drag wakes is of interest to a wide variety of applications in geophysics and engineering, such as the wakes of mountains, sea-mounts, windfarms, and buildings. For axisymmetric bodies, the classical self-similar scaling law formulated by Swain (1929) has long been widely accepted (Tennekes and Lumley 72) and used to describe the wakes of spheres (Bevilaqua and Lykoudis 78) and slender bodies (Pao & Lin 73). However, recent experiments have cast doubt on the classical decay law (Bonnier and Eiff 02; Nedic, Vassilicos, and Ganapathisubramani 13), indicating more rapid wake spreading and decay than the classical result. Here, new laboratory data from a dimpled sphere (Re = 50,000) are compared with various theoretical explanations for non-classical self-similar wake decay.

8:39AM H40.00004 Spreading and decay of axisymmetric drag wakes: experiment, D. CURTIS SAUNDERS, SCOTT WUNSCH, Johns Hopkins University — The spreading and decay of drag wakes is of interest to a wide variety of applications in geophysics and engineering, such as the wakes of mountains, sea-mounts, windfarms, and buildings. For axisymmetric bodies, the classical self-similar scaling law formulated by Swain (1929) has long been widely accepted (Tennekes and Lumley 72) and used to describe the wakes of spheres (Bevilaqua and Lykoudis 78) and slender bodies (Pao & Lin 73). However, results from recent experiments have cast doubt on the classical decay law (Bonnier and Eiff 02; Nedic, Vassilicos, and Ganapathisubramani 13), indicating more rapid wake spreading and decay than the classical result. Here, new laboratory data from a dimpled sphere are presented. The Reynolds number (50,000) exceeds previous experiments, and the dimples alleviate issues with Strouhal vortex shedding. Particle Imaging Velocimetry (PIV) data in both along-track and cross-track configurations are used to illustrate various aspects of the wake decay. The observed wake decays at a greater rate than suggested by the classical result, and extends the self-similar decay to further downstream distances than previously reported.

8:52AM H40.00005 Investigation of Large-Scale Coherent Structures in Flow past a Sphere for Scale-Resolving Simulations (SRS), CHEHTA KAMBLE, FREDDIE WITHERDEN, SHARATH GIRIMAJI, Texas A&M University — Success of Scale-Resolving Simulations (SRS) depends on accurately reproducing the large-scale structures in complex turbulent flows. Yet, no clear procedure exists for a quantitative assessment of the fidelity of these flow structures. The objective of this work is to develop such an assessment framework in the context of wake flow past a sphere which is chosen due to its inherent complexity and three-dimensional features. Numerical computations at different degrees of physical resolution are performed using the Partially-averaged Navier-Stokes (PANS) bridging-SRS method. Proper Orthogonal Decomposition (POD) is employed for a quantitative analysis of the computed large-scale structures in the wake. Most energetic structures resolved at different physical resolution are extracted and compared. Furthermore, the dominant structures are also qualitatively examined using the iso-surfaces of Q-criterion, i.e., second invariant of velocity gradient tensor. These results are compared to Direct Numerical Simulations (DNS) studies to establish the efficacy of the PANS-SRS simulations in capturing the large-scale vortical structures.

9:05AM H40.00006 Direct measurements of the mode B instability in the wake of a two-dimensional blunt trailing edge, BRADLEY GIBEAU, SINA GHAEMI, University of Alberta — Recent work has revealed contradictory results regarding the secondary instabilities that appear in the wake of a two-dimensional body with a blunt trailing edge (BTE). A linear stability analysis at Reynolds numbers near transition predicted that the aspect ratio (AR) of the body, i.e. the ratio of chord length to body thickness, dictates the secondary instabilities that exist in the wake. These authors predicted that a mode B’ instability will appear in the wake for AR $> 7.5$, in contrast to the mode B that exists in the wakes of cylindrical bodies (AR = 1). There are experimental results in the literature to support the existence of mode B’. However, the indirect analysis techniques used to show its existence have recently been found to be inadequate. We match the AR of 12.5 used in the studies that reported mode B’ and also extend to a large AR of 46.5 and conduct particle image velocimetry measurements to unambiguously characterize the secondary instability in the wake. Mode B is found to be present in the wake for all Reynolds numbers investigated (Re(h) = 2600 to 26000 where h is the thickness of the BTE), suggesting that AR does not play an important role in the formation of the secondary instability. No evidence of mode B’ was found.
Stratified near wake of a slender body

Jose Luis Ortiz-Tarin, Sheel Nidhan, Sutantu Sarkar, University of California San Diego — Motivated by applications in submersible hydrodynamics, the effect of density stratification on the wake of a slender body is numerically investigated. Large-eddy simulations with an immersed boundary method are employed to simulate the flow past a 6:1 prolate spheroid at zero angle of attack. The Reynolds number based on the minor-axis diameter is fixed to \( Re_D = UD/\nu = 10^5 \), and different levels of stratification measured by \( Fr = U/ND \) are compared: \( Fr = 2 \) which is close the critical value of Froude number \( (Fr_c = 6/\pi ) \) where the turbulence suppression in the wake is maximal; a moderately stratified case \( (Fr = 4) \) and the unstratified case \( (Fr = \infty) \). A trip placed near the nose of the body is used to force laminar-turbulent transition and the wake is simulated to \( 25D \). Preliminary results that include the evolution of the boundary layer, mean velocity and turbulence statistics will be presented.

Office of Naval Research

High Reynolds Number Stratified Turbulent Sphere Wake Measurements

Kenneth Kalumuck, Alan Brandt, Johns Hopkins U. Applied Physics Lab — There are virtually no data on wake turbulence for Reynolds numbers \( (Re) \) of one million or larger, above the drag crisis, even though aircraft and ships operate at \( Re \sim 10^6 \). This study seeks to characterize the near-field of a stratified wake at large \( Re \sim 2 \times 10^5 - 1.4 \times 10^6 \), by towing a large diameter \( (\sim 0.5 \text{ m}) \) sphere through a thermally stratified lake and a thermally stratified large salt water tow tank. The stratification produced BV frequencies, \( N \), up to 0.07 s\(^{-1}\) with Froude numbers \( F = U/ND \geq 15 \). Three component turbulent velocity and temperature measurements were obtained using Acoustic Doppler Velocimeters (ADVs) and an array of fast response thermistors at various downstream distances. Wake turbulence characteristics including rms velocity and temperature fluctuation spectra, dissipation, integral scales, and Reynolds stresses are presented. Velocity and temperature power spectra exhibit clear -5/3 slopes while velocity co-spectra and transverse spectra exhibit -7/3 slopes over order-of-magnitude ranges in wavenumber, which are generally not clearly evident for lower \( Re \) laboratory experiments.

Office of Naval Research Subsurface Hydrodynamics Program

Spectral Proper Orthogonal Decomposition of Flow Around a Disk in Homogeneous and Stratified Fluids

Sheel Nidhan, University of California San Diego, Karu Chongsiripinyo, Chulalongkorn University, Sutantu Sarkar, Oliver Schmidt, University of California San Diego — The elucidation of coherent structures and quantification of their role is a cornerstone of research in turbulent flows. Proper orthogonal decomposition (POD) introduced by Lumley to turbulent flows is a technique which can be used to identify the coherent structures that optimally capture the fluctuation kinetic energy of the flow. In this study, a space-time variant of POD, namely spectral POD (SPOD) is used to study the dynamics of the wake behind a disk at \( Re = 50,000 \). Two cases, a stratified flow with \( Fr = U/ND = 2 \) that evolves into a regime of strongly stratified turbulence (SST) and an unstratified case with \( Fr = \infty \), are analyzed. Preliminary results for \( Fr = \infty \) show that the wake fluctuation energy is dominated by azimuthal modes, \( m = 0, 1, \text{and } 2 \). As the wake evolves downstream beyond \( x/D \approx 60 \), \( m = 2 \) (at Strouhal number, \( St = 0 \)) becomes dominant while the \( m = 1 \) mode (at \( St = 0.136 \)) dominates in the near wake. At \( Fr = 2 \), the energy captured in the most energetic POD mode increases downstream, indicating the increasing importance of coherent modes in the stratified wake as it evolves downstream. In further analysis of modal dynamics, the Reynolds stresses carried by the dominant modes and the intermodal energy transfer among these modes will be quantified.

Office of Naval Research

Direct Numerical Simulation of a Turbulent Wake behind a Body of Revolution at \( Re_D = 5000^3 \)

Fengrui Zhang, Yulia Peet, Ronald Adrian, School for Engineering of Matter, Transport and Energy, Arizona State University — This study is concerned with the numerical investigation of a three-dimensional wake behind a body of revolution via Direct Numerical Simulations. Direct Numerical Simulations with the Reynolds number \( Re_D = 5000 \) based on the bluff body diameter are performed using a high-order spectral-element solver Nek5000. The focus of the study is on characterizing the wake asymmetries and low-frequency behavior observed in previous experimental studies with similar bluff body models. Modal analysis will be presented to show the wake dynamics in both the near-wake and the far-wake region.

The current work is supported by NSF CBET grant 1707075.

Vertical Flux of Mean Flow Kinetic Energy in the Near-Wake of a Marine Propeller in the Presence of External Turbulence

Luksa Luznik, Bennitt Hermse, United States Naval Academy — This experimental study examines the spatial evolution of the three-bladed marine propeller model \((D=13\text{cm})\) near wake in the early stages from the immediate wake behind the propeller up to 7 propeller diameters downstream for two inflow conditions: one with imposed external turbulence with \( 7\% \) intensity and integral scale compared to propeller blade geometry, and the second one with clean inflow conditions as a reference. Resulting Reynolds number is \( Re_D \approx 4.7 \times 10^5 \). All components of the vertical fluxes of mean flow kinetic energy are calculated from available three-dimensional PIV data and compared for the two inflow conditions. Influence of external turbulence on the wake instability process of mutual induction is examined and it was found that external turbulence enhances tip-vortex roll up mechanism resulting in earlier breakdown of individual vortices into small scale turbulence. Conditional sampling is performed to analyze the mechanisms of mean flow kinetic energy transport and it was found that outward interactions and sweep events contribute the most to the vertical transfer of mean flow kinetic energy from the inner wake to the free stream.

Funded by Office of Naval Research
10:23AM H40.00012 Do ribs help succulents to cope with aerodynamic loads in their natural environment?1
OLEKSANDR ZHDANOV, ANGELA BUSSE, University of Glasgow — Ribs on cacti stems play an important role for their survival not only in terms of water storage and evaporation control but also in attenuating high wind loads. Studies of flow past circular cylinders with many ribs inspired by the Saguaro cactus, which is native to the North American deserts, showed a reduction in drag and amplitude of unsteady force fluctuations compared to the smooth circular cylinder. In the Eastern Hemisphere some succulents have independently developed a similar plant structure as cacti but with a low number of ribs. If aerodynamics is one of the driving factors behind this convergent evolution, shapes with a low number of ribs should show similar benefits. In the present work, we experimentally investigated the aerodynamics of a cylinder with four ribs inspired by the succulent Euphorbia Abyssinica. As its outer shape is approximately square, i.e. non-circular, the aerodynamic coefficients show a strong dependence on its orientation with respect to the mean flow. The experimental results are compared to the square cylinder at the same angular orientations and to results obtained from large-eddy simulations. The optimal orientation of this shape where aerodynamic loads are minimised is also discussed.

1This work is supported through LKAS scholarship scheme by the University of Glasgow

Monday, November 25, 2019 8:00AM - 10:36AM – Session H41 Uncertainty Quantification  6c - Catherine Gorle, Stanford

8:00AM H41.00001 Uncertainty quantification in CFD simulations of natural ventilation to support designing experiments for model validation1  
CHEN CHEN, CATHERINE GORLE, Stanford University — Natural ventilation can significantly reduce building energy consumption, but the variability in the boundary and operating conditions makes the robust design of an HVAC system challenging. In previous studies, a computational framework using an integral model and a computational fluid dynamics (CFD) model with uncertainty quantification (UQ) was used to predict the volume-averaged indoor air temperature during night-time ventilation in Stanford’s Y2E2 building. Comparison to point-wise building sensor measurements indicated that spatial variability in the temperature field is non-negligible. Hence, the sensor measurements might not be representative of the volume-averaged temperature. The objective of the present study is to use CFD simulations with UQ to design an experiment that optimizes temperature sensor placement to (1) obtain measurements that are representative of the volume-averaged temperature, and (2) support further analysis of the spatial variability in the temperature field for validation of CFD results. The presentation will discuss the methodology used to account for the variability in the boundary and initial conditions, the design of experiments based on the results, and the use of the measurements for validation of the model results.

1This work is funded by the Center for Integrated Facility Engineering at Stanford University.

8:13AM H41.00002 Uncertainty quantification of the scale determination in steady RANS modeling for turbulence with large-scale structures1  
ZENGRONG HAO, CATHERINE GORLE, Stanford University, WIND ENGINEERING LAB TEAM — The scale determining variable (e.g. energy dissipation rate $\epsilon$, frequency scale $\omega$, or length scale $l$) is notoriously difficult to model and thus a major source of uncertainties in RANS simulations for turbulence. Particularly in the turbulence with large-scale structures that usually yield highly non-equilibrium energy spectra, a steady RANS simulation using a single modeled scale can result in considerable errors, especially for energy levels. In this report, we present a dual-scale, three-equation steady RANS approach to quantify the uncertainties originating in the scale determination in a model for the turbulence with large-scale structures (LS). The uncertainty quantification (UQ) approach is developed in three steps. First, we obtain the equations for the energies of LS and quasi-equilibrium small-scale turbulence (ST) from a triple decomposition, and employ a traditional model for the dissipation rate of ST energy. Second, under the limit condition where LS and ST have identical scales, we derive the form of the energy transfer rate (ETR) from LS to ST via a conceptual analysis with the 'return-to-isotropy' evolution of Reynolds' stresses. Last, a marker with a single uncertain parameter is designed to identify the regions potentially bearing LS (particularly for those in large separation regions), and is used to suppress the ETR. Applications of this UQ approach in several cases show its capability of encompassing the real energy levels in the separation regions.

1IWT-110068

8:26AM H41.00003 Estimating model-form uncertainty for multi-scale systems  
JIN-LONG WU, TAPIO SCHNEIDER, ANDREW STUART, California Institute of Technology — Global climate models (GCMs) are widely used to simulate climate change. However, their predictions have large uncertainties, arising primarily in subgrid-scale (SGS) parameterizations for globally unresolved small-scale processes. Work to quantify parametric uncertainties in these parameterizations is underway, using methods from data analysis and machine learning. Quantifying structural or model-form uncertainties is more challenging. Here we propose a non-parametric approach to model structural errors that uses Gaussian processes and ensemble Kalman sampling and that respects physical constraints (e.g., energy conservation). We illustrate how this approach can be used to quantify model-form uncertainties in low-dimensional multi-scale systems and with an idealized GCM. The results demonstrate that this approach allows us to go beyond merely calibrating model parameters toward quantifying model uncertainty more broadly.

8:39AM H41.00004 Uncertainty quantification for RANS simulations of variable density flows  
AAASHWIN MISHRA, ZHU HUANG, JAN HEYSE, GIANLUCA IACCARINO, Center for Turbulence Research, Stanford University, TIMOTHY CLARKE WALLSTROM, DAVID SHARP, Los Alamos National Laboratory — Variable density turbulent flows are widely encountered in a variety of natural phenomena and industrial applications, from the Jovian atmosphere and its Giant Red Spot to nuclear applications such as Inertial Confinement Fusion. RANS models and specifically eddy viscosity closures are widely used for investigations into variable density turbulence. However, RANS models have shortcomings in accounting for the effects of anisotropy, rotation, streamline curvature. This is further exacerbated by the forced alignment of the heat flux with the temperature gradient and the relationship between eddy diffusivity and viscosity. These assumptions lead to significant errors and uncertainties in turbulent model predictions for such flows. In this investigation, we outline the application and extension of tensor perturbations to estimate the uncertainties in variable density turbulent flows, deriving uncertainty estimates for the Besnard-Harlow-Raupenzahn (BHR) model. This is carried out for a variety of flows including tilted rocket rigs, variable density turbulent jets, etc. The selected cases show that extension of tensor perturbation can be utilized for uncertainty estimation for predictions of variable density turbulent flows.
Order Models for multifidelity uncertainty quantification, JIAN-XUN WANG, LUNING SUN, HAN GAO, University of Notre Dame, SHAOWU PAN, (W911NF-15-1-0562) and by ARO Young Investigator Program Award (W911NF-19-1-0444).

Flow fields, hence predictive simulations of flow instabilities. brake system input parameters in addition to the investigation of forward uncertainty propagation from the available data and model into the corresponding forward and backward uncertainty quantifications are also performed. This study leads to the Bayesian inference of the stochastic complex-geometry capturing, and cost-efficient direct numerical simulation of the fluid flow. Given available experimental (PIV) data, the rotating cylinder, fully filled with water, arrives at rest within a short time-period from a constant rotational speed. We employ spectral element method to perform highly-accurate, nature of vorticity dynamics, arising from the uncertain initial/ boundary/ topological conditions. The rotating cylinder, subject to an imperfect/ random rotational brake system is modeled and simulated to better understand the stochastic nonlinear behavior.

For modeling closure, we provide an inferred eddy viscosity using two different strategies. The main test case is the flow over an airfoil, with the QoI taken as the lift and/or drag. The parameter space is taken as the angle-of-attack and the Reynolds number. Here, the accuracy of the frozen eddy viscosity assumption for different parameters is assessed.

Model form uncertainty arises from physical assumptions made in constructing models either to reduce the physical complexity or to model physical processes that are not well understood. Understanding these uncertainties is important for both quantifying prediction uncertainty and understanding the source and nature of model errors. Data-based methods for model form uncertainty quantification (UQ) are limited by the availability of data, and, in turbulence, data is often limited by Reynolds number or geometry. In contrast, physics-based UQ seeks to analyze the model form uncertainty by leveraging the physical assumptions inherent in these models so that it can be extrapolated beyond available data. In this work, an "implied models" approach is developed and applied to Reynolds stress modeling. In the approach, a model error transport equation is derived from the fundamental governing equations by taking the difference between the exact Reynolds stress equation and the equation implied by a particular model form. Budgets of the model error transport are analyzed to better understand the sources of error in RANS models, focusing on the relative contributions from the Boussinesq hypothesis and the specific form of the eddy viscosity in canonical turbulent flows.

Many realistic science and engineering applications require complex high-fidelity (HF) simulations for the characterization of the system’s response, in combination with large numbers of random parameters that need to be propagated through these HF simulations. In these cases, a single fidelity approach for UQ becomes intractable due to the extreme cost of resolving both deterministic and stochastic errors. Multifidelity strategies have been introduced to alleviate this issue by fusing information from simulations with varying levels of fidelity, in order to obtain estimators that preserve HF statistics at much lower overall cost. This is typically accomplished through a priori definition of a sequence of model physics or discretizations of varying accuracy and expense. Less attention has been dedicated to the automatic generation of low-fidelity models using data from a small number of HF simulations. In this work, we focus on the case in which low-fidelity models are data-driven using HF samples/snapshots, with initial emphasis on projection-based reduced-order models.

We investigate how the accuracy and certainty of the quantities of interest (QoIs) of canonical wall-bounded turbulent flows are sensitive to various numerical parameters and time averaging. The scale-resolving simulations are performed by Nek5000, an open-source high-order spectral-element code. Different uncertainty quantification (UQ) techniques are utilized in the study. Using non-intrusive polynomial chaos expansion, portraits of error in the QoIs are constructed in the parameter space. The uncertain parameters are taken to be the grid spacing in different directions and the filtering parameters. As a complement to the UQ forward problems, global sensitivity analyses are performed with the results being quantified in the form of Sobol indices. Employing Bayesian optimization based on Gaussian Processes, the possibility of finding optimal combinations of parameters for obtaining QoIs with a given target accuracy is studied. To estimate the uncertainty due to time averaging, the use of different techniques such as classical, batch-based and autoregressive methods is discussed and suggestions are given on how to efficiently integrate such techniques in large-scale simulations. Comparisons of the certainty aspects between high-order and low-order codes (OpenFOAM) are given.

Funding by the Excellerator CoE is acknowledged.

Numerical simulations on fluid dynamics problems can often be computationally prohibitive for most realistic and many-query applications, and developing a cost-effective surrogate model of great practical significance. Deep learning (DL) has shown new promises for surrogate modeling due to its capability of handling strong nonlinearity and high dimensionality. However, the off-the-shelf DL architectures fail to operate when the data becomes sparse, which is often the case in most parametric fluid dynamics problems since each data point in the parameter space requires an expensive numerical simulation. In this paper, we provide a physics-constrained DL approach for surrogate fluid modeling without relying on any simulation data. Specifically, a structured deep neural network (DNN) architecture is devised to enforce the boundary conditions, and the Navier-Stokes equations are used to drive the training. Numerical experiments are conducted on a number of vascular flows and forward propagation of uncertainties in fluid properties and domain geometry is studied as well. The results show excellent agreement on the flow field and propagated uncertainties between the DL surrogate approximations and the first-principle numerical simulations.

This work was supported by the AFOSR Young Investigator Program Award (FA9550-17-1-0150), and partially by MURI/ARO (W911NF-15-1-0562) and by ARO Young Investigator Program Award (W911NF-19-1-0444).

9:44AM H41.00009 Surrogate Modeling for Fluid Flows Using Physics-Constrained, Label-Free Deep Learning, JIAN-XUN WANG, LUNING SUN, HAN GAO, University of Notre Dame, SHAOWU PAN, University of Michigan, Ann Arbor — Numerical simulations on fluid dynamics problems can often be computationally prohibitive for most realistic and many-query applications, and developing a cost-effective surrogate model of great practical significance. Deep learning (DL) has shown new promises for surrogate modeling due to its capability of handling strong nonlinearity and high dimensionality. However, the off-the-shelf DL architectures fail to operate when the data becomes sparse, which is often the case in most parametric fluid dynamics problems since each data point in the parameter space requires an expensive numerical simulation. In this paper, we provide a physics-constrained DL approach for surrogate fluid modeling without relying on any simulation data. Specifically, a structured deep neural network (DNN) architecture is devised to enforce the boundary conditions, and the Navier-Stokes equations are used to drive the training. Numerical experiments are conducted on a number of vascular flows and forward propagation of uncertainties in fluid properties and domain geometry is studied as well. The results show excellent agreement on the flow field and propagated uncertainties between the DL surrogate approximations and the first-principle numerical simulations.

9:57AM H41.00010 Stochastic Simulation of Flow Instabilities in a Rotating Cylinder, S. HADI SEYEDI, ALI AKHAVAN-SAFAEI, JOHN FOSS, MOHSEN ZAYERNOURI, Michigan State University — A rotating cylinder subject to an imperfect/ random rotational brake system is modeled and simulated to better understand the stochastic nonlinear nature of vorticity dynamics, arising from the uncertain initial/ boundary/ topological conditions. The rotating cylinder, fully filled with water, arrives at rest within a short time-period from a constant rotational speed. We employ spectral element method to perform highly-accurate, complex-geometry capturing, and cost-efficient direct numerical simulation of the fluid flow. Given available experimental (PIV) data, the corresponding forward and backward uncertainty quantifications are also performed. This study leads to the Bayesian inference of the stochastic brake system input parameters in addition to the investigation of forward uncertainty propagation from the available data and model into the flow field, hence predictive simulations of flow instabilities.
Orthogonal (DO) method and Bi-Orthogonal (BO) method for highly ill-conditioned cases. In particular, we demonstrate that the presented real-time reduced-order modeling of transient stochastic systems for the purpose of uncertainty propagation. We present a closed-form evolution equation for a low-rank time-dependent basis that evolves with the dynamics. An on-the-fly reduction of the dynamical system is then obtained by projecting the full-dimensional system onto the low-rank basis. The presented method is compared against the state-of-the-art Dynamically Orthogonal (DO) method and Bi-Orthogonal (BO) method for highly ill-conditioned cases. In particular, we demonstrate that the presented method preserves the accuracy of the solution and the shape of the modes as the system passes through a low eigenvalue phase. The results for several demonstration cases are presented, including linear advection equation, stochastic Burgers’ equation and stochastic flow in a channel.

least-squares Petrov-Galerkin (LSPG) ROM (equipped with hyper-reduction) has demonstrated the ability to significantly reduce simulation costs while retaining high levels of accuracy on a range of problems including subsonic CFD applications. This work presents the first application of LSPG to hypersonic CFD problems including the Blottner sphere and HiFIRE experimental flight vehicle. This shows the ability of LSPG ROMs while retaining high levels of accuracy.

1NL is managed and operated by NTESS under DOE NNSA contract DE-NA0003525
2Coauthors: Francesco Rizzi, NexGen Analytics; Micah Howard, Sandia National Laboratories; Jeff Fike, Sandia National Laboratories; Kevin Carlberg, Sandia National Laboratories

10:23AM H41.00002 Dynamically bi-orthonormal formulation for stochastic partial differential equations
1, PRERNA PATIL, HESSAM BABAE, University of Pittsburgh — A new method is presented for a real-time reduced-order modeling of transient stochastic systems for the purpose of uncertainty propagation. We present a closed-form evolution equation for a low-rank time-dependent basis that evolves with the dynamics. An on-the-fly reduction of the dynamical system is then obtained by projecting the full-dimensional system onto the low-rank basis. The presented method is compared against the state of the art Dynamically Orthogonal (DO) method and Bi-Orthogonal (BO) method for highly ill-conditioned cases. In particular, we demonstrate that the presented method preserves the accuracy of the solution and the shape of the modes as the system passes through a low eigenvalue phase. The results for several demonstration cases are presented, including linear advection equation, stochastic Burgers’ equation and stochastic flow in a channel.

Least-Squares Petrov–Galerkin Reduced-Order Models for Hypersonic Flight Vehicles
PETER BLOM, PATRICK BLONIGAN, Sandia National Laboratories, FRANCESCO RIZZI, NexGen Analytics, MICAH HOWARD, JEFF FIKE, KEVIN CARLBERG, Sandia National Laboratories — High-speed aerospace engineering applications rely heavily on computational fluid dynamics (CFD) models for design and analysis due to the expense and difficulty of flight tests and experiments. This reliance on CFD models necessitates performing accurate and reliable uncertainty quantification (UQ). However, it is very computationally expensive to run CFD for highly grid resolution cases. Additionally, UQ methods are many-query problems requiring many runs with a wide range of input parameters. One way to enable computationally expensive models to be used in such many-query problems is to employ projection-based reduced-order models (ROMs) in lieu of the (high-fidelity) full-order model. In particular, the least-squares Petrov-Galerkin (LSPG) ROM (equipped with hyper-reduction) has demonstrated the ability to significantly reduce simulation costs while retaining high levels of accuracy on a range of problems including subsonic CFD applications. This work presents the first application of LSPG to hypersonic CFD problems including the Blottner sphere and HiFIRE experimental flight vehicle. This shows the ability of LSPG ROMs to significantly reduce computational costs while maintaining high levels of accuracy in computed quantities of interest.

1American Chemical Society, Grant number 59081-DN19

Monday, November 25, 2019 8:00AM - 10:36AM –
Session H42 Boundary Layers: Multi-scale Roughness and Secondary Flows

8:00AM H42.00001 Time-varying secondary flows in turbulent boundary layers over surfaces with spanwise heterogeneity
1, DEA WANGSAWIAYA, KEVIN KEVIN, DANIEL CHUNG, IVAN MARUSIC, NICHOLAS HUTCHINS, The University of Melbourne — Secondary flows form in turbulent flow over surfaces with spanwise heterogeneity. In this study, we use surfaces comprised of spanwise alternating smooth and rough (P-36 grit sandpaper) strips to investigate the behaviour of these secondary flows for various strip widths. PIV measurements are performed on the wall-parallel plane above surfaces with various strip widths $S$ ($0.3 \leq S/\delta \leq 3.6$), where $\delta$ is the spanwise-averaged boundary layer thickness. We find that, when $S/\delta \approx 1$, these secondary flows not only strengthen but also exhibit a pronounced unsteadiness. This unsteadiness is consistent with a fluctuating from side-to-side of large-scale streaks with a streamwise wavelength of 3-4f.

8:13AM H42.00002 Turbulent wall flows over spanwise-heterogeneous surfaces: non-periodic deviation from Reynolds-averaged flow patterns
WILLIAM ANDERSON, UT Dallas — Turbulent flows respond to bounding walls with a predominant spanwise heterogeneity – that is, a heterogeneity parallel to the prevailing transport direction – with formation of Reynolds-averaged turbulent secondary flows. These secondary flows constitute manifestation of Prandtl’s secondary flow of the second kind: driven and sustained by spatial heterogeneities in the Reynolds (turbulent) stresses. Results from large-eddy simulations and complementary experimental measurements of flow over spanwise-heterogeneous surfaces are shown: the resultant secondary cell location is clearly correlated with the surface characteristics, which ultimately dictates the Reynolds-averaged flow patterns. However, results also show the potential for instantaneous sign reversals in the rotational sense of the secondary cells. This is accomplished with probability density functions and conditional sampling. In order to further this, a base flow representing the spanwise rolls is introduced. Upon substitution of the base flow into the streamwise momentum and streamwise vorticity transport equations, and via use of a vortex forcing model, we assess phase-space evolution (orbit) of the resulting system of ordinary differential equations. The system resembles the Lorenz system, but the forcing conditions differ intrinsically. Nevertheless, the system reveals that chaotic, non-periodic trajectories are possible for sufficient inertial conditions.

8:26AM H42.00003 Extending the restricted nonlinear model to flow over rough surfaces with spanwise heterogeneities
1, XIAOWEI ZHU, BENJAMIN MINNICK, DENNICE GAYME, Johns Hopkins University — The restricted nonlinear (RNL) model has shown success in accurately predicting statistical features of smooth wall-turbulence. The RNL equations are obtained by decomposing the Navier-Stokes equations into a streamwise-averaged component and streamwise-varying perturbations, and then restricting the nonlinearity in the perturbations. The resulting dynamics are supported by a small number of streamwise varying modes ($k \times w$ wave numbers) interacting with the streamwise constant mean flow. In this work, we extend the RNL modeling paradigm to rough walls by first determining the streamwise wave number support that correctly predicts the log-law behavior associated with rough wall flows. We demonstrate that the parametrization can be obtained analogously to the smooth-wall case, where it was shown that the particular streamwise varying modes required to correctly reproduce low-order statistics and spectra at low and moderate Reynolds numbers are associated with the outer-layer peak of the surrogate dissipation spectra. Comparisons of DNS and RNL simulations over riblets at various spanwise spacing using an immersed boundary method indicate that the properly parameterized rough-walled RNL model reproduces low-order statistics and spanwise energy spectra reminiscent of the DNS data.

1This work is supported by Office of Naval Research (N000141712049).
2Corresponding Author: dennice@jhu.edu
8:39AM H42.00004 Effect of roughness texture on transient, accelerating channel flows. SAI CHAITANYA MANGAVELLI, JUNLIN YUAN, GILES BRERETON, Michigan State University — The effect of surface roughness on non-equilibrium, accelerating wall turbulence is studied using direct numerical simulation of transient periodic channels. The smooth-wall base case is compared with two irregular rough surfaces: i) a small-wavelength sand-grain roughness and ii) a multiscale turbine-blade roughness. The flow is accelerated from a bulk Reynolds number of 3000 to 12000 in a short time interval, rendering the rough-wall flows transitionally-rough to fully-rough. The smooth wall undergoes reverse transition towards a laminar-like state with quasi-1D turbulence, being replaced by a new equilibrium state. In contrast, near a rough wall a more isotropic Reynolds stress tensor and a higher friction coefficient are observed. This is mainly due to the fast responses of the form-induced Reynolds-stress production and pressure work to the increased shear, both contributing significantly to higher Reynolds-stress isotropy. As the characteristics of the form-induced fluctuations are important for these mechanisms, the roughness texture determines the initial rate of turbulence response. Results show that the roughness geometry is important in a non-equilibrium turbulence over a rough wall.

1Financial support from ONR (N00014-17-1-2102) is acknowledged

8:52AM H42.00005 Secondary flow structure development in multi-scale rough wall turbulent boundary layer. NASEEM ALI, JULIANTA BOSSUYT, BIANCA VIGGIANO, Portland State University, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, JOHAN MEYERS, KU Leuven, RAUL CAL, Portland State University — Fluid motions in the narrow slits between the cubical roughness elements are resolved. We find pairs of relatively small-sized counter rotating rollers triggered by a Kelvin-Helmholtz type instability. Measurements are used to characterize changes in the mean profile and turbulence statistics, and to test for the emergence of spanwise-coherent disturbances. As Reynolds number increases, the turbulence statistics change, and the flow undergoes a large transition from the low-Reynolds stress state to a high-Reynolds stress state. The new transition mechanism is found to be a combination of large-scale ejections that carry energy from the boundary layer to the outer region. The results are relevant to the prediction of drag and boundary layer development over urban surfaces and roughness transition in the presence of secondary flows.

9:05AM H42.00006 Turbulent boundary layer development over a step change in multiscale roughness. JOHN LAWSON, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — We examine the development of the turbulent boundary layer across a step-change transition between four different three-dimensional, multiscale, cuboidal roughness surfaces. We collected wind-tunnel measurements of the flow above the canopy in spanwise stereo-PIV planes at 72 streamwise locations up- and downstream of the transition, as well as direct measurements of the skin-friction drag immediately downstream of the transition. This extensive dataset allows us to quantify the development of the resultant internal boundary layer in terms of its mean velocity profile, Reynolds and dispersive stresses, using true repeating-unit averaging. In addition, we compare measurements of surface drag to estimates from a recently-developed morphometric drag prediction model (Yang et al. 2016). Our findings are relevant to the prediction of drag and boundary layer development over urban surfaces and roughness transition in the presence of secondary flows.

1EPSRC EP/P021476/1

9:18AM H42.00007 On the interaction of streamwise vortices with surface textures. SAIKISHAN SURYANARAYANAN, DAVID GOLDSMITH, The University of Texas at Austin, GARRY BROWN, Princeton University, ALEXANDRE BERGER, EDWARD WHITE, Texas AM University — Recent studies (Suryanarayanan et al., AIAA 2017-4417, AIAA 2018-3077) have demonstrated the possibility of mitigating roughness induced transition using surface textures that can dissipate the streamwise vortices responsible for the lift-up effect. In order to identify the optimal surface texture geometry, a fundamental investigation into the effect of surface textures on the evolution of a streamwise vortex in a boundary layer is performed. This study involves a combination of scaling arguments (of the enstrophy flux transport equation) and direct numerical simulations of a single streamwise vortex in a laminar boundary layer with different (2D) surface textures. The results are analyzed from a vorticity dynamics point of view. The effects of local flow acceleration and enhanced dissipation are separately studied using appropriate synthetic simulations. Simple models are suggested and connections are made to previous work on the evolution of perturbations over wavy walls. Supporting wind tunnel experiments that study the development of streamwise vortices generated by micro vortex generators in the presence of surface textures will be presented. Applications to cross flow instability and other technologically relevant scenarios will be discussed.

1Supported by AFOSR grant FA9550-19-1-0145

9:31AM H42.00008 Measurements in turbulent boundary layers over designed anisotropic porous materials. CHRISTOPH Efstathiou, MITUL Luhar, University of Southern California — This talk describes the design and testing of anisotropic porous media with directional resistance as a means to control the development of the turbulent boundary layer. A high-resolution stereolithographic 3D printer is used to manufacture a substrate with higher streamwise than wall-normal permeability ($\phi_{xy} > k_{xx}$). Such streamwise-preferential materials have demonstrated potential for passive turbulence suppression and drag reduction. Measurements are made using a Particle Image Velocimetry system in a high-speed wind tunnel in a large-scale water channel facility. The friction Reynolds number is varied between $Re_f \approx 200$ to $Re_f \approx 2000$. Measurements are made using a time-resolved Particle Image Velocimetry system and a 2-component Laser Doppler Velocimeter mounted on a precision traverse. These velocity measurements are used to characterize changes in the mean profile and turbulence statistics, and to test for the emergence of spanwise-coherent rollers triggered by a Kelvin-Helmholtz type instability.

1This work was supported by the Air Force Office of Scientific Research under AFOSR grant no. FA9550-17-1-0142 (Program Manager: Dr. Gregg Abate).

9:44AM H42.00009 Cube-roughened surfaces at high and extreme packing densities. HAOSEN XU, XIANG YANG, The Penn State University — We study turbulent channel flows with rough walls comprised of aligned arrays of cubic roughness elements at high surface coverage densities ($\lambda$). Specifically, we carry out direct numerical simulations (DNS) at $\lambda = 0.5 \sim 0.9$. Fluid motions in the narrow slits between the cubic roughness elements are resolved. We find pairs of relatively small-sized counter rotating vortices in between and above the cubes at moderately high packing densities, i.e., $\lambda = 0.5 \sim 0.7$. These vortices are replaced by large ones above each cube when surface coverage is high ($\lambda \leq 0.8$). Last, we will compare our DNS to wall-modeled LES, which is a more affordable tool than DNS at high Reynolds numbers. For the flows considered, WMLES compare reasonably well with the DNS, although noticeable differences in the roughness occupied layer is found.
10:10AM H42.00011 Three-dimensional measurements of flow structure and turbulence around a pair of cubic roughness elements embedded in the inner part of a turbulent channel. JIAN GAO, JOSEPH KATZ, Johns Hopkins University — The 3D flow and turbulence around a pair of roughness cubes embedded in the inner part of a turbulent channel flow (Re = 2500) are measured using microscopic tomographic holography accelerated using GPU-based algorithms. The cube height, a = 1 mm, is 90 wall units. The cubes decrease the near-wall velocity and generate secondary flows as far as 3a upstream. Horseshoe vortices form at the front surface, and propagate asymmetrically around the cubes, generating secondary vortices along the side surfaces. Each cube and its near wake are engulfed by a vortical canopy dominated by wall-normal vorticity along the sides and spanwise vorticity above the cube. Merging of the “tip vortices” developing along the cube upper edge, horseshoe and secondary vortices occurs downstream at locations that decrease with the spacing. They form a large streamwise vortex behind each cube, which rotates in the same direction as the inner leg of the horseshoe vortex. With decreasing spacing, the flow accelerates faster between cubes, but also decelerates faster in the near wake. Streamwise velocity fluctuations of 40% of the freestream velocity and negative Reynolds shear stress develop near the front upper corner of the cube. The turbulence remains high around the entire surface and near wake.

1Fundied by ONR.

10:23AM H42.00012 Experimental flow characteristics over different area density roughness. Kwonho Park, HEECHANG LIM, Pusan National University — In this study, we aim to observe the flow and surface pressure characteristics over a variety of roughness pattern in a boundary layer wind tunnel. For each roughness element, the cuboid shape was chosen and size 50x50x50mm3. The atmospheric boundary layer (hereafter, ABL) has an important role of exchange and transfer in momentum flux at ground level. The roughness sub-layer (RS), the lower part of ABL, is usually 2 times thicker than the building height, and the urban canopy layer (UCL) was the same as the building height. Therefore, this study performed experimentally the generation of boundary layer in a wind tunnel and observed the flow characteristics of boundary layer and the surface pressure distribution around each roughness. Besides, to observe the effect of the roughness pattern, changes in area density of surface roughness were made by 11, 17, and 25%, of boundary layer in a wind tunnel and observed the flow characteristics of boundary layer and the surface pressure distribution around each roughness. In addition, the surface pressure at top face becomes higher as area density increases, whereas the pressure on the back remains almost constant regardless of area density. In addition, the surface pressure at top face becomes higher as area density increases.

Monday, November 25, 2019 11:00AM - 11:35AM — Session J01 Invited Talk: Impact of Wettability on Multiphase Flow and Granular Mechanics: Experiments, Modeling and Theory 6b - Gareth McKinley, MIT

11:00AM J01.00001 Impact of Wettability on Multiphase Flow and Granular Mechanics: Experiments, Modeling and Theory. RUBEN JUANES, Massachusetts Institute of Technology — Many natural subsurface processes involve the interaction between multiphase flow and deformation of porous media like rocks and soils. Examples include hydraulic fracturing, induced seismicity from fluid injection, and subsidence from groundwater extraction, just to name a few. In some cases, capillary effects can control surface tension force, and in some cases there is a fundamental role in the fluid-solid interaction. Here, we report some recent observations on how, in these cases, the flow and deformation are strongly modulated by wettability, which is the relative affinity of each fluid to the solid making up the porous medium. These observations are surprising and intriguing, but a mechanistic explanation has heretofore remained elusive. Here, we present a fully-coupled dynamic model of granular mechanics and multiphase flow at the pore scale, which explicitly incorporates the impact of wettability. This mechanistic model allows us to explore the rich emerging behavior as a function of different parameters, such as capillary number, contact angle, initial packing density, and grain rigidity. Beyond the suggestive predictions of pattern formation, the model also hints at the origin of the transitions between patterns. We reconcile the rich behavior we observe in terms of a jamming transition, which opens a promising way to understand novel aspects of wet granular systems.

Monday, November 25, 2019 11:00AM - 11:35AM — Session J02 Invited Talk: Unsteady aerodynamic response of rigid wings in gust encounters 6c - Beverly McKeon, Caltech

11:00AM J02.00001 Unsteady aerodynamic response of rigid wings in gust encounters. ANYA R. JONES1, University of Maryland, College Park — Controlled flight in unsteady environments presents a challenge that has gained interest in recent years. Unsteady flight conditions exist in many scenarios including urban areas, airwaves, and extreme weather. In cases such as these, large force transients on wings and other lifting surfaces occur due to rapid variations in effective angle of attack resulting in flow separation and the formation of large-scale vortices. The growth and motion of these vortices can have a large impact on the resulting force transient and recovery, and often requires advanced control either locally via flow control or more globally at the vehicle level. Current research efforts focus on the effect of large wind gusts that result in changes to the relative flow that are of the same order of magnitude as the freestream flow. In these large-amplitude gust encounters, flow separation is significant, so the classical linear solution for the flow does not apply. Separated shear layers emanating from the wing tend to roll up into leading and trailing edge vortices that are shed into the wake. The formation and motion of these vortices are characterized via a series of canonical experiments in an attempt to better understand their contribution to aerodynamic forcing.

1This research was supported by the Japan Society for the Promotion of Science (JSPS), KAKENHI(18K04471).
This work has been funded by the National Science Foundation and has benefitted greatly from the support of the GFD program at the Woods Hole Oceanographic Institution.
1:58PM L01.00002 Transport of gas mixtures in a Knudsen pump with specular and diffuse walls, TOBIAS BAER, STEFFEN HARDT, Institute for Nano- and Microfluidics, TU Darmstadt, Germany — Gas flow in Knudsen pumps is induced by thermal gradients in channels or pores when the mean free path of the gas molecules is comparable to the geometric feature size. While thermal transpiration is often associated with flow along pores with a net axial temperature gradient, flow can also be induced by imposing a temperature difference across the channel, when suitably structured walls induce a periodic but non-mirror-symmetric temperature profile along the channel. One such arrangement, inspired by the Crookes radiometer, consists of placing an array of plates with different reflection properties on their opposing sides along a channel. By direct simulation Monte Carlo (DSMC) we investigate the transport of gas mixtures along such channels, focusing on the discrimination of individual species in the mixture by molecular size and mass during flow due to temperature, composition and pressure gradients. As Knudsen pumps do not possess any moving parts, they lend themselves for operation in low-maintenance situations such as for feeding gases to sensor surfaces. The observed discrimination of different gas species may thus adversely or favorably influence the overall detection characteristic of such a combined device.

2:11PM L01.00003 Soot chemical pathways under elevated pressures in co-flow ethylene flames, SUO YANG, DEZHI ZHOU, HONGYUAN ZHANG, University of Minnesota — Internal combustion engines and gas turbines are operating under high pressures. One of the major concerns in high pressure combustion is its high soot yield, which was found in many sooting flame experiments. Specifically, the maximum soot volume fraction increases with the increasing pressure, with the dependence of soot on pressure weaker as pressure is further increased. To explain the dependence of soot on pressure, most of the experiments attribute this phenomenon to higher temperature, steeper precursor concentration gradients, and increased gas density. However, an exact and comprehensive mechanism behind this phenomenon, from a chemical kinetics perspective, is still elusive. In this study, a series of pressurized (from 1 to 16 atm) co-flow ethylene diffusion flames are simulated with detailed finite rate chemistry. The soot evolution is described by the bivariate Hybrid Method of Moments (HMM). The experimental maximum soot volume fraction in the flames are reproduced by the simulations. Most importantly, the Global Pathway Analysis (GPA) is conducted to reveal the dominance and sensitivity of soot chemical pathways under elevated pressures.

2:24PM L01.00004 Influence of Temperature and Dilution on Final Soot Nanostucture, JUSTIN DAVIS, IGOR NOVOSSELOV, University of Washington — Morphological evolution of nascent to mature combustion-generated particles is of interest due to changes in particle optics, chemical composition, and their effect on human health. In this work, the nanostructure of primary soot particles is investigated using argon dilution in laminar ethylene, ethane, and methane flames. A co-flow reactor oriented downwards allows for precise control on combustion conditions due to increased flame stability brought on by competing buoyant and convective forces. The dilution is varied from 0% to 90% by volume to investigate particle formation in temperature ranges from 1500 to 1950 K. High-resolution TEM displays different levels of particle maturity, from young soot with minimal order to mature particles with a core-shell nanostructure. A novel image processing algorithm helps to quantify differences in soot nanostructure, indicating the molecular weight of PAHs is similar for young and mature soot. However, molecular curvature decreases, suggesting the driving factor of soot maturity is a reduction in steric hindrance due to carbonization kinetics at high flame temperatures. These results can be used for validation of soot modeling approaches and to improve the understanding of structural changes as soot particles traverse the flame front.

2:37PM L01.00005 ABSTRACT WITHDRAWN — 

2:50PM L01.00006 Experimental investigation of solid fuel smoldering using 3D X-ray computed tomography with gas-phase temperature measurements, EMERIC BOIGNE, MATTHIAS IHME, Stanford University — Smoldering is a regime of combustion that is characterized by low temperatures, slow rates of fuel consumption, and the absence of a flame. Recent research investigations on smoldering have focused on applications within confined environments such as peat fires, subsurface wildfires, and remediation of soil contamination. The present work reports on the use of 3D X-ray computed tomography to experimentally investigate smoldering in confined configurations. By temporally resolving the surface recession of solid fuel materials, the local consumption rates are extracted at the submillimeter scale. By diluting the ambient flow with Krypton, an inert X-ray tracer, the X-ray measurements enable simultaneous estimations of the 3D gas-phase temperature field. First, a configuration within an over-ventilated environment is considered for validation. Then, more complex fuel-configurations are examined, exhibiting spatial fuel heterogeneities and different fuel materials. Effects of the spatial heterogeneities on the smoldering process are discussed using the temporally-resolved 3D measurements. The simultaneous measurements of local consumption rate and gas-phase temperature allow further investigations of state-of-the-art models of smoldering.

3:03PM L01.00007 Experimental Investigation of Combustion in Porous Media Burners with Tailored Matrix-Structure Using Additive Manufacturing, PRIYANKA MUNHUNTHAN, SADAF SOBHANI, EMERIC BOIGNE, DANYAL MOHADDES, MATTHIAS IHME, Stanford University — Porous Media Burners (PMBs) have been shown to enable enhanced burning velocities, extend flammability limits, and lower emissions. In this study, the viability of 3D printing complex ceramic structures is examined, and the performance of a novel functionally graded PMB design is investigated. Three different alumina matrix foams were designed and printed using lithography-based ceramic manufacturing. The flame stability, temperature profiles, and pressure drop for each burner are examined. X-ray computed tomography is used to characterize the printed matrix structure and to obtain important geometric information about pore diameter and porosity profiles. This work builds upon previous experimental investigations into graded PMBs, and the results demonstrate the feasibility of using additive manufacturing for tailoring PMB topologies to achieve specific system requirements.

Monday, November 25, 2019 1:45PM - 3:16PM — Session L02 Multiphase Flows: Modeling and Theory II 2B - Duan Zhang, Los Alamos National Laboratory
1:45PM L02.00001 High-Fidelity Simulation of a Rotary Bell Atomizer with Electrohydrodynamic Effects, VENKATA KRISSHNA, MARK OWKES, Montana State University — Rotary Bell Atomizers (RBA) are extensively used as paint applicators in the bell competitive industry. Atomization of paint is achieved by a bell cup rotating at high speeds (40k-60k RPM) and in the presence of a background electric field. Current estimates report a maximum paint transfer efficiency of only 60%. The atomization process in an RBA affects the droplet size and velocity distribution which subsequently control the transfer efficiency and surface finish quality. Moreover, optimal spray parameters used in industry are often obtained from expensive trial-and-error methods. In this work, a computational approach is used to simulate three-dimensional near-cup flows (mainly primary and secondary breakup) using a high-fidelity volume-of-fluid transport scheme that includes the effects of Electrohydrodynamics (EHD). This work involves the use of two meshes, one to solve the Navier-Stokes equations in the atomization region and a second on a larger region to solve for the electric field with realistic boundary conditions. This research aims to develop a cost-effective method to investigate the influence of various flow characteristics on the atomization and breakup processes of the liquid.

1:58PM L02.00002 An Eulerian-Eulerian CFD Study of Surface Wetting at Different Richardson Numbers in Dispersed Oil-Water Pipe Flow, JAKOB ROAR BENTZON, JENS HONORE WALther, Technical University of Denmark — An Eulerian-Eulerian two-phase CFD model has been employed to investigate the variation of dispersion and liquid holdup in the horizontal section of wells used for oil production under operating conditions with water-cut ranging from 25-75%. The employed model uses a 5-gamma droplet distribution model to estimate droplet sizes based on statistical moments with breakup and coalescence models. The model has been validated against experimental measurements with good accuracy on phase distribution but challenges obtaining correct droplet sizes. Consequently, the model is used to study different flow conditions, namely at varied Richardson numbers through changing the Froude number and the Atwood number separately. From the results, it is observed that similarly to what is seen in the Kelvin-Helmholtz instability, a higher Richardson number reduces mixing of the interface and thus decreases the rate of dispersion. Eight differences in the results from varying the Atwood number and the Froude number to same Richardson numbers are observed on both dispersion and liquid holdup.

2:11PM L02.00003 Thermal and Fluid Dynamics of Snow vs. Rain at the Air-Water Interface, MEHDI VAHAB, Kourosh ShoELE, Florida State University, DAVID MURPHY, University of South Florida — The effects of precipitation in the forms of rain, snow, and hail are studied using computational methods for multimaterial/multiphase systems. The comparison of single droplet impacts shows the phase-dependency (a liquid or solid droplet) of the momentum and energy transfer to the surface and body of water. The depth of penetration and the resultant vertical flow are found highly dependent on the phase-change rate of the droplet. Multiple models of snow particles with systematic geometrical complexities are tested for the role of shape in air pocket entrapment at the impact. The sensitivity of the single droplet impact dynamics is also studied by varying droplets size and temperature, and are used for investigation of the precipitation events with the average ensemble effects of multiple droplet impacts. The size and velocity of the droplets are set based on the observed size/velocity distribution of frequent rain, hail, or snow events. A predictive model is developed to predict the changes in the water surface and body energy content following each precipitation event.

2:24PM L02.00004 Assessment of surfactant models with applications to crude oil-water coalescence or aggregation modeling, SHAOLIN MAO, Engineering Department, University of Jamestown, ND — We study surfactant models with focus on crude oil — sea water aggregation/coalescence analysis and modeling. Undesired crude oil-water emulsion exists in most crude oil production and recovery process; therefore, costly process such as destabilizing of oil-water emulsion in petroleum engineering is required from reservoirs, wellbores, to wellheads and transport of crude oil. This research is limited to physics of coalescence or aggregation of water in oil mixing (crude oil is the dominant phase). Parameter studies will be conducted in this work to assess surfactant models, elucidate the factors which influence the aggregation/coalescence of crude oil-sea water mixing process. Numerical simulation is based on commercial CFD tool such as ANSYS FLUENT by using population balance model (PBM) coupling with CFD solvers. Experimental observation and comparison will also be given.

2:37PM L02.00005 Manipulating gas-assisted atomization by inlet gas turbulence, DELIN JIANG, YUE LING, Baylor University, DANIEL FUSTER, STEPHANE ZALESKI, Sorbonne Universites, GRETAIR TRYGGVASON, Johns Hopkins University — In an airlift atomization process, the destabilization and breakup of a liquid jet is assisted by a co-flowing high speed gas stream. Recent experiments and simulations show that the inlet gas turbulence has a strong impact on the interfacial instability development near the nozzle exit and also the liquid breakup downstream. In this study we will systematically investigate the effect of inlet gas turbulence on the interfacial instability and spray formation through high-fidelity simulation. The gas-liquid interface is resolved by a momentum-conserving volume-of-fluid method. A digital filter approach is used to generate temporally and spatially correlated velocity fluctuations at the gas inlet. The dominant frequency and spatial growth rate of the mixing layer are observed to increase significantly with the inlet gas turbulence intensity. The dominant frequency predicted by simulations are in a good agreement with the experimental data. Spatial-temporal viscous linear instability analysis is also conducted using the eddy-viscosity model. The simulation results also reveal the change of dominant breakup mechanism when gas inlet turbulence is present.

2:50PM L02.00006 Sloshing and its effects on thermal mixing in aircraft fuel tanks, CONNOR ROHMANN-SHAW, DUNCAN BORMAN, MARK WILSON, University of Leeds — Modern aircraft designs are becoming ever more complex, with higher demands on their performance capabilities. As the thermal loads from the airframe, engine, electrical systems etc. increase, one of the key challenges is thermal management. A solution is to use the fuel as a coolant; fuel is used as a heat sink for components around the aircraft and then recirculated back into the fuel tanks. To fully optimise this method however we must expand upon existing thermal management models by better understanding the thermal mixing process internal to the fuel tanks, with particular attention given to the role of fuel sloshing. The multi-phase Volume of Fluid method of free-surface tracking is used to predict fluid motion, which we couple to the thermal flow field using the Boussinesq approximation. Using these numerical simulations, we investigate the effect that sloshing has on thermal mixing, allowing us to inform future design considerations. Experiments undertaken using thermocouples to measure the evolution of the temperature field in a turbulent free-surface flow will also be presented.

1Supported by the EPSRC [grant EP/L01615X/1]
Effect of inertial forces on constitutive behaviors of the RVE granular model

We are grateful to the financial support from Los Alamos Machine Learning Program.
2:37PM L03.00005 A proprioceptive robotic swimmer. — MEDERIC ARGENTINA, JESUS SANCHEZ RORIDGUEZ, CHRISTOPHE RAUFASTÉ, Université Cote d’Azur, Nice France — In the context of fish locomotion, Lighthill proposed in 1971 his elongated body theory to predict the swimming gait [1]. Since his seminal article, the swimming velocity has been related to the fin kinematics (tail-beat amplitude and frequency) through the Strouhal number [2], which is found to be constant over 6 decades of Reynolds number for natural swimmers [3]. This result can be accounted for by a simple force balance, but the selection mechanism of the fin kinematics is still poorly understood. Here, we propose that the swimming locomotion might be driven by a proprioceptive feedback, which determines the tail-beat amplitude and frequency. A robotic fish has been built to test this mechanism and we will present our last results.


2:50PM L03.00006 Empirical generalizations for swim speed and endurance of sturgeon species — CHRISTOS KATOPODIS, Retired, LU CAI, Institute of Hydroecology, Ministry of Water Resources and Chinese Academy of Sciences, Wuhan 430079, China, RICHARD GERVAIS, Freshwater Institute, Fisheries and Oceans Canada, Winnipeg R3T 2N6, Canada — Swim performance data of various fish species ranging from prolonged (low) to burst (high) speeds are useful for both theoretical hydrodynamics and ecophysical applications, such as fish passage. Commonly such data are collected with experimental tests, are limited to fish speed and endurance for a small percentage of species, and are insufficient for variables such as tailbeat frequency. Limited data are available for the Sturgeon species, especially for burst speeds, and dimensionless variables use them more effectively. Robust regressions for several groups of species, as well as sturgeon, are obtained with a dimensionless fish speed, expressed in the form of a fish Froude number which does account for fish length. Regressions improve with dimensionless variables, including fish speed expressed as a fish Reynolds number, compared to traditional measures such as body lengths/s. Such regressions demonstrate swim performance similarity in sturgeon and other fish groups, offer empirical data generalizations, data collection strategies, and help validate biomimetic simulations.

3:03PM L03.00007 Impact of hydrodynamics of porous and non-porous structures on upstream fish passage performance1 — STEPHANIE MUELLER, ELIZABETH FOLLETT, CATHERINE WILSON, PABLO OURO, JO CABLE, Cardiff University — Flow visualization and velocity measurements using ADV were used to assess upstream and wake flow characteristics of non-porous and porous structures made of wooden dowels. Upstream of the structure, flow becomes diverted towards the bed with higher downwards vertical velocities for the non-porous case. A turbulent wall jet formed beneath, showing higher flow acceleration with decreasing porosity, leading to stronger turbulent momentum exchange through the shear layers in the wake. Behind the non-porous structure a larger recirculation area formed. At the downstream edge of the structure, turbulent kinetic energy (TKE) was larger for the non-porous case, with highest levels found approximately at mid-structure height. In the porous cases, two peaks in TKE occurred at the trailing edge of the lowest dowel, due to inter-dowel wake effects. Swimming behavior observations of rainbow trout (Oncorhynchus mykiss) revealed fish spent time beneath the structure, likely to avoid the high momentum jet. With increasing structure porosity, fish preferred to swim in the structure’s wake due to reduced mean velocities and turbulent fluctuations near the bed. These observations indicate that porous and non-porous structures create heterogeneous habitats, influencing fish behavior.

1funded by the UK Engineering and Physical Sciences Research Council (EP/L016214/1)

Monday, November 25, 2019 1:45PM - 3:16PM — Session L04 Focus Session: Probing Multi-scale Flows by Coarse-graining 203 - Nicholas T. Ouellette, Stanford

1:45PM L04.00001 Gaining “insight” by blurring one’s “sight”. — HUSSEIN ALUIE, University of Rochester — Coarse-graining (CG), which is equivalent to observing flows through eyeglasses of varying strength, provides a powerful yet intuitive approach to analyze and understand multiscale flows. This is because it allows for resolving the dynamics simultaneously in scale and in space, at every instant in time, and without requiring assumptions of homogeneity or isotropy, making it ideal to probe complex non-canonical flows that are unsteady, inhomogeneous, anisotropic, and multi-phase. With examples from plasma, oceanic, and variable-density flows, I will illustrate some of the advantages CG has over traditional tools such as Reynolds averaging, structure functions, and Fourier analysis. I will discuss a new method based on CG to extract the spectrum, including that of non-quadratic quantities such as kinetic energy in variable density flows, self-consistently, which the Fourier spectrum and the wavelet spectrum cannot. I will also discuss the emergence of new invariants in MHD and in compressible turbulence which can be unraveled by the proper scale-decomposition based on CG. Finally, I will present a generalization of CG to spherical domains, allowing for the analysis of scale-processes in oceanic flows from satellites.

1:58PM L04.00002 Enhanced Spectral Transfer in Weakly Mixing Regions of a Turbulent Flow1 — LEI FANG, NICHOLAS OUELLETTE, Stanford University, SANJEEVA BALASURIYA, University of Adelaide — Scale-to-scale spectral energy flux is a hallmark of turbulent flows. The geometric alignment of small-scale turbulent stress and large scale rate of strain leads to the net flux of energy from small scales to large scales in 2D turbulence. We have found, however, that the instantaneous alignment between these two tensors is surprisingly weak, and thus that the spectral transport of energy is inefficient. In our experimental work, we have shown that the strain rate is much better aligned with the stress at times in the past, suggesting that the differential advection of the two is responsible for the inefficient spectral transfer. Based on this understanding, we developed a tool to specifically look for weakly mixing regions in the flow based on the linearity this implies, which we term Linear Neighborhoods (LN). We demonstrate that these LN’s are computable in real data using experimental measurements from a 2D turbulent flow. Consistent with our previous results, we find that that the spectral energy flux behaves differently inside the LN’s, where the spectral energy flux is more efficient. Our results add additional support to the conjecture that turbulent flows locally tend to transport energy and momentum in space or between scales but not both simultaneously.

1U.S. National Science Foundation under Grant No. CMMI-1563489
2:11PM L04.00003 Baroclinic Energy Transfer in the Ocean

The role of baroclinicity, which arises from the misalignment of pressure and density gradients, is well-known in the vorticity equation, yet its role in the kinetic energy budget has never been obvious. Ref. [1] has recently shown that baroclinicity appears naturally in the kinetic energy budget after carrying out the appropriate scale decomposition. Here, we extend the coarse-graining decomposition to study this process within the shallow water model and apply it to numerical simulations as well as satellite data. [1] A. Lees and H. Aluie, Fluids 4, 92 (2019).

1Supported by NASA grant number 80NSSC18K0772

2:24PM L04.00004 Eigenframe alignment dominates scale-to-scale energy fluxes in turbulence

In large-eddy simulation, the cascade is cut off at a certain scale and the transfer term to smaller scales is modeled. This transfer term depends on the eigenvalues of the turbulent stress and resolved strain-rate tensors and the Euler angles between their eigenframes. In a typical Smagorinsky closure, two assumptions are made: these angles are set to zero and the stress eigenvalues are modeled as scaled version of the stress. We show that it is the Euler angles that have a much larger effect on the transfer in terms of the directionality of the cascade and its efficiency.

2:37PM L04.00005 Filter-width and Atwood number effects in filtered homogeneous variable density turbulence

We investigate Atwood number (A) and filter width (w) dependence in filtered DNS of buoyancy-driven homogeneous variable density turbulence, where density differences affect mixing and turbulence, and we discuss implications for modeling. We show that statistics and budgets of filtered fields transition smoothly between DNS and RANS fields and budgets, and we discuss these transitions in the context of flow length scales. At small w, filtered fields tend to DNS fields and the large-scale flow kinetic energy (k1) budget tends to the total kinetic energy (k2) budget, at large w, filtered fields approach RANS fields and k1 approaches the mean kinetic energy (k0) budget. At intermediate w, the k1 budget has dissipation and pressure-dilatation work terms from the k1 budget, a mean pressure gradient term from the k0 budget, a production term from both the k1 and k0 budgets, and work by residual stresses against the filtered shear $\varepsilon_k$, which tends to zero at both limits. Work by mean pressure gradients and by $\varepsilon_k$ exhibit density dependent back-scatter: at high A, $\varepsilon_k$ back-scatter occurs mainly in liquid. Statistics of filtered fields, normalized by their RANS counterparts, smoothly and monotonically vary between 0 and 1 as w varies from dx to domain size, and the dependence on w is different for different quantities.

2:50PM L04.00006 Data-driven coarse-grained modeling for polymers

Iao Pan, University of Wisconsin-Madison — We present a data-driven coarse-graining method to establish coarse-grained (CG) modeling for polymers, which conserves both static and dynamic properties of the fine-grained (FG) system. The dynamics of the CG system is governed by the generalized Langevin equation (GLE) derived via the Mori-Zwanzig formalism, by which the CG variables can be directly linked to the statistics of FG data. The effect of unresolved degrees of freedom on the kinetics of polymers can be captured by the non-Markovian stochastic dynamics in GLE, where the memory kernel is determined from the FG data. To circumvent the difficulty of directly solving the GLE with memory term and colored noise, we exploit the equivalence between the non-Markovian dynamics and Markovian dynamics in an extended space. To this end, the CG system is supplemented with auxiliary variables that are coupled linearly to the momentum and among themselves, subject to uncorrelated Gaussian white noise. For several different polymer systems in melts or in solution, we demonstrate that the established CG modeling can reproduce both static and dynamic properties of the reference FG system.

1NSF grant CMMI-1761068

3:03PM L04.00007 Is vortex stretching the main cause of the turbulent energy cascade?

Andrew Bragg, Duke University, Maurizio Carbone, Polytechnic University of Turin — While the dominant idea is that in 3D turbulence, the energy cascade occurs through the process of vortex stretching, evidence for this is debated. In the framework of the Karman-Howarth equation, we derive a new result for the average flux of kinetic energy between two points in the flow. The result shows that vortex stretching is in fact not the main contributor to the average energy cascade; the main contributor is the self-amplification of the strain-rate field. We emphasize the need to correctly distinguish and not conflate the roles of vortex stretching and strain-self amplification in order to correctly understand the physics of the cascade, and also resolve a paradox regarding the differing role of vortex stretching on the mechanisms of the energy cascade and energy dissipation rate. Direct numerical simulations are used to confirm the results, as well as provide further results and insights on vortex stretching and strain-self amplification at different scales in the flow. Interestingly, the results imply that while vortex stretching plays a sub-leading role in the average cascade, it may play a leading order role during large fluctuations of the energy cascade about its average behavior.

Monday, November 25, 2019 1:45PM - 3:16PM — Session L05 Shock-Boundary Layer Interactions

1:45PM L05.00011 Low-frequency unsteadiness in a shock wave boundary layer interaction

Roi Baidya, Sven Scharnowski, Matthew Bross, Christian J. Kähler, Bundeswehr University Munich — Large field-of-view (FoV) particle image velocimetry experiments are conducted in the vicinity of a shock wave boundary layer interaction (SWBLI) at Mach 2. The current FoV covers up to 30 boundary layer thicknesses, comprising of upstream and downstream regions relative to the SWBLI, thereby allowing the turbulent boundary layer and shock to be simultaneously captured. The relationship between the boundary layer features and the instantaneous shock location is directly quantified, with the aim of better understanding the mechanisms responsible for oscillation of the reflected shock. Simultaneous wall-pressure measurements indicate that the low-frequency fluctuation arising from the oscillating shock foot is not necessarily an independent phenomenon from the turbulent features entering the SWBLI region and interact with the shock. Instead, a large scale separation between their dominant time-scales is due to dampening of high-frequency content beyond the critical frequency of the globally unstable mode occurring at frequencies that are orders of magnitudes slower than the dominant frequency of the very-large-scale velocity features.

1This work is supported by the Priority Programme SPP 1881 funded by the German Research Foundation (Deutsche Forschungsgemeinschaft — DFG).
1:58PM L05.00002 Shock-wave boundary layer interactions in an engine intake with a spectral/hp element method.\textsuperscript{1} , GIACOMO CASTIGLIONI, FRANCESCO MONTOMOLI, SPENCER J. SHERWIN, Imperial College London — During take off and climbing, i.e. at high angle of attack and high mass flow rate through the engine, the flow on the upper surface of a nacelle bottom lip can develop a region of supersonic flow. The supersonic pocket terminates with a near normal shock-wave which interacts with the incoming boundary layer leading to a shock-wave boundary layer interaction (SWBLI). The aim is to assess the capability of discontinuous Galerkin spectral element methods in conjunction with an artificial viscosity shock capturing method to predict the onset of SWBLI unsteadiness in complex geometries relevant to industrial applications. Here it is simulated an experimental rig recently investigated by Coschignano et al. The rig is designed around a section of the bottom dead center of a real 3D intake lip, the geometry downstream of the lip is arbitrary and resembles that of an airfoil. Under resolved direct numerical simulations are performed at design conditions which are characterized by an angle of incidence of 23\textdegree and a free stream Mach number $Ma = 0.435$ resulting in a closed, shock-induced, boundary layer separation; the Reynolds number based on lip thickness overlaps the lowest one available experimentally ($Re_x = 4 \times 10^5$).

\textsuperscript{1}The authors would like to thank EPSRC for the computational time made available on the UK supercomputing facility ARCHER via the UK Turbulence Consortium EP/R029326/1 and by the High Performance Computing facilities at Imperial College London.

2:11PM L05.00003 Shockwave Boundary Layer Interaction Control Using External Forcing from Nanosecond Repetitively Pulsed Dielectric Barrier Discharge , RAVICHANDRA JAGANNATH, LALIT RAJENDRAN, GEORGE SCHMIDT, TANBO ZHOU, SALLY BANE, Purdue University — The interaction between shock waves and turbulent boundary layers is an interesting phenomenon in high speed aerodynamics. The presence of a shock wave creates an adverse pressure gradient leading to boundary layer separation. This interaction causes unsteadiness in the flow where the separation bubble oscillates at low frequency and the shock oscillates at broadband frequency. The cause of this unsteadiness still remains unanswered. Some results show an upstream influence, while others have shown the downstream separation bubble causing the unsteadiness. This uncertainty in the unsteadiness mechanism makes it difficult to find an ideal location for an active flow control actuator. In the past, plasma jets, localized arc filaments and quasi-DC discharges have been used to control the shock unsteadiness. In this study, a nanosecond repetitively pulsed surface dielectric barrier discharge is used on a 25\textdegree compression corner in Mach 2.5 flow to create an external perturbation in the upstream turbulent boundary layer ahead of the shock wave. The goal is for the external perturbation to be convected downstream and influence the shock oscillation. A combination of schlieren visualization and wall pressure measurements is used to investigate the effect of plasma actuator on shock unsteadiness. The actuator is placed at different locations upstream of the shock to identify the optimal location. The plasma frequency is also varied to identify the optimal frequency for controlling the shock unsteadiness.

2:24PM L05.00004 Analysis of the effect of elasticity on Shock Turbulent Boundary Layer Interaction\textsuperscript{1} , JONATHAN HOY, IVAN BERMEJO-MORENO, University of Southern California — The effect of elasticity on the unsteady behavior of a shock-turbulent boundary layer interaction (STBLI) is investigated through the use of a coupled fluid structure interaction (FSI) solver which incorporates a wall-modeled large eddy simulation (WMLES) finite-volume flow solver, an undamped finite-element solid mechanics solver, and a mesh deformation solver based on a fictitious spring-system which calculates the change in flow domain mesh geometry as it is deformed by the solid domain. The FSI solver is validated through comparison to the experimental work of Willems et al (2016) and to the LES simulations of Pasquariello et al (2015). In these cases, a supersonic flow at Mach 3 is deflected downward by 20 degrees to create an oblique shock wave that impinges on the turbulent boundary layer developed over a flexible elastic panel (with a Reynolds number of 205,000 based on the boundary layer thickness upstream of the interaction). Parametric studies are then performed to investigate the impact of panel vibration on the STBLI and separation bubble dynamics, varying the panel natural frequencies and shock strength.

\textsuperscript{1}This work has been partially supported by NASA

2:37PM L05.00005 Mean and Unsteady Characteristics of Swept SBLIs\textsuperscript{1} , SATHYAN PADMANABHAN, JORGE CASTRO MALDONADO, JAMES THREADGILL, JESSE LITTLE, University of Arizona — An experimental study has been conducted on swept impinging oblique shock turbulent boundary layer interactions (SBLIs) generated by 12.5\textdegree shock generators at various sweep angles in Mach 2.3 flow. Mean and unsteady features are examined using oil flow visualization, mean pressure measurements, and high bandwidth pressure transducers. Mean results show the flow is fully separated, and for higher sweep angles, exhibits a spanwise growth of the interaction, displaying a conical behavior away from the inception region. Unsteady pressure measurements reveal low-frequency unsteadiness along the separation shock foot, with reduced amplitude in comparison to similar strength unsept interactions. Separation shock motion is coherent along the span in lower frequencies while spanwise traveling ripples present in the shock foot accelerate from the root towards the tip over the range of 15-25\% of $U_{\infty}$. Minor correlation is observed between shock motion and pressure disturbances directly upstream in the incoming boundary layer, but this becomes negligible when offset in the spanwise direction. This suggests that shock rippling is not driven by upstream disturbances, but instead associated with a time scale inherent to the interaction.

\textsuperscript{1}Air Force Office of Scientific Research (FA9550-15-1-0430)

2:50PM L05.00006 Structure and Unsteadiness of Shock-foot and Separation in a Swept SBLI at Mach 2\textsuperscript{1} , LEON VANSTONE, MUSTAFA MUSTA, NOEL CLEMENS, The University of Texas at Austin — This study examined the structure and unsteadiness of a 3D shock wave / boundary layer interaction (SBLI) generated by a swept compression ramp in a Mach 2 flow. Fast response PSP was used to examine the behavior of the separation shock foot unsteadiness and high frequency PIV data was used to examine the separation line unsteadiness. Shock-foot unsteadiness was broadband, having significant low- ($St_{0.01}$), mid- ($0.01; St_{0.10}$), and high-frequency ($St_{0.10}$) content. Generally, the shock foot contained lower frequency content in comparison to the separation line. In a low-frequency band the behavior of the shock foot was similar to that of the separation line. The mid- and high-frequency band shock-foot unsteadiness was not characteristically similar to the separation location. Many studies examine SBLI unsteadiness using only a single characterising feature (shock foot, separation line, etc.). Our results showed that the shock and separation lines have different characteristic lines. Hence, different features within the SBLI give different characteristic impressions of unsteadiness.

\textsuperscript{1}This work is sponsored by the AFOSR under grant FA9550-14-1-0167 with Ivett Leyva as the program manager.
Carbon Dioxide Material Synthesis Reactor, COURTNEY OTANI, ELIZABETH RASMUSSEN, JOHN KRAM-...

1 bench-top measurements of known concentrations, and implemented on a co-flowing jet injector facility. Preliminary results demonstrating the

diametrically are measured using visible-light absorption spectroscopy at various distances from the injector. The diagnostic is characterized using

between 1-Napthol and Diazotized Sulfanilic acid. The concentrations of mono- (primary) and di-azo (secondary) reaction products averaged

the chemical reaction products in complex turbulent environments. We present a novel in-line spectroscopic measurement to study the reaction

necessitating the need for better injector designs for turbulent control. The relevant experimental studies frequently require the ability to measure

products ('reaction yield') involving consecutive-competitive reactions have long been known to depend on characteristics of turbulent mixing,

industrial processes involve mixing-limited reaction chemistry in highly turbulent and complex environments. The proportions of reaction

as three others are analyzed using computational fluid dynamics (CFD) to determine the flow characteristics of various geometries. Velocity

optimizing reactor conditions for uniform and controllable crystallization growth. Therefore, the commonly used counter-current design as well

flow sCO2 material synthesis reactor. Modeling the transient and multiphase mixing between sCO2 and precursor materials gives insight into

simulations of chemical reactions in the presence of two-dimensional, laminar shear flow using both the one-dimensional Eikonal approximation,

which assumes fronts are thin curves with an inherent burning speed, as well as the two-dimensional advection-reaction-diffusion equation, which

captures the dynamics of reactant concentration through both space and time. We find that the key factor in improving mixing efficiency is to

maximize the absolute velocity difference in the flow. We have also developed an analytical solution to model these front dynamics and have

successfully predicted the final front shape and growth rate for six unique flow profiles.

We gratefully acknowledge support from The Dow Chemical Company under the University Project Initiative.

---

Monday, November 25, 2019 1:45PM - 3:16PM —

Session L06 Reactive Mixing 205 - Douglas Kelley, University of Rochester

1:45PM L06.00001 Comparing laminar flows for efficient reactive mixing, DANIEL TROYET-SKY, THOMAS D. NEVINS, DOUGLAS H. KELLEY, University of Rochester — As a chemical reaction occurs, a front is formed at the boundary between reacted and non-reacted regions. We wish to quantify the efficiency of reactive mixing by a variety of shear flows by understanding how fronts evolve over time. A more efficient flow causes the reacted region to grow more quickly. To do this we have performed numerical simulations of chemical reactions in the presence of two-dimensional, laminar shear flow using both the one-dimensional Eikonal approximation, which assumes fronts are thin curves with an inherent burning speed, as well as the two-dimensional advection-reaction-diffusion equation, which captures the dynamics of reactant concentration through both space and time. We find that the key factor in improving mixing efficiency is to maximize the absolute velocity difference in the flow. We have also developed an analytical solution to model these front dynamics and have successfully predicted the final front shape and growth rate for six unique flow profiles.

1:58PM L06.00002 Reactive mixing in a shear flow, EMILIE GUILBERT, CHRISTOPHE ALMARCHA, Aix Marseille University, HENRI LHUSSIERS, BLOEN METZGER, CNRS, EMMANUEL VILLERMAUX, Aix Marseille University — We investigate the interaction between chemical reaction and diffusion on a moving substrate and by means of an original chemical reaction between two transparent reactants (fluorescin and potassium ferricyanide) which produce a fluorescent product in water (fluorescein) with tunable kinetics. A blob of fluorescin in a cell filled with potassium ferricyanide is advected in a simple shear flow. The blob deforms into an elongated strip diffusing from its borders at a shear-dependent rate. Depending on the ratio of the mixing time to the reaction time (Damköhler number $Da$), several regimes are identified: When $Da > 1$, the product concentration increases linearly in time at a rate controlled by the reaction only. The mixing-controlled decay rate of the final product concentration after the reaction is completed is described as well.

2:11PM L06.00003 Flow Characteristics of Mixing in Continuous Flow Supercritical Carbon Dioxide Material Synthesis Reactor, COURTNEY OTANI, ELIZABETH RASMUSSEN, JOHN KRAM-LICH, IGOR NOVOSSELOV, University of Washington — Supercritical carbon dioxide ($scO2$) is being increasingly employed for advanced clean energy research including turbomachinery, electronics cooling, and more recently material synthesis. Understanding the fluid mechanics that dictate optimum operation is key to wide-spread application. The research presented here focuses on the mixing section of a continuous flow $scO2$ material synthesis reactor. Modeling the transient and multiphase mixing between $scO2$ and precursor materials gives insight into optimizing reactor conditions for uniform and controllable crystallization growth. Therefore, the commonly used counter-current design as well as three others are analyzed using computational fluid dynamics (CFD) to determine the flow characteristics of various geometries. Velocity and temperature contour plots and profiles are presented to identify regions with high vorticity, large temperature gradients, areas of stagnation, and boundary layer effects. Comparison of these results highlight the attributes and inefficiencies of different mixing section geometries.

2:24PM L06.00004 In-line spectroscopic diagnostics to investigate mixing-limited consecutive-competitive reaction systems, GOKUL PATHIKONDA, MICHAEL AHMAD, MUSTAFA USTA, Georgia Institute of Technology, IRFAN KHAN, Dow Company, CYRUS AIDUN, DEVESH RANJAN, Georgia Institute of Technology — Many complex industrial processes involve mixing-limited reaction chemistry in highly turbulent and complex environments. The proportions of reaction products (‘reaction yield’) involving consecutive-competitive reactions have long been known to depend on characteristics of turbulent mixing, necessitating the need for better injector designs for turbulent control. The relevant experimental studies frequently require the ability to measure the chemical reaction products in complex turbulent environments. We present a novel in-line spectroscopic measurement to study the reaction between 1-Napthol and Diazotized Sulfanilic acid. The concentrations of mono- (primary) and di-azo (secondary) reaction products averaged diametrically are measured using visible-light absorption spectroscopy at various distances from the injector. The diagnostic is characterized using bench-top measurements of known concentrations, and implemented on a co-flowing jet injector facility. Preliminary results demonstrating the effect of viscosity gradients on reaction yield is presented. It was noticed that an increase in the viscosity gradients in the reaction environments (owing to disparity in viscosity of inlet liquids) significantly alters the selectivity of the overall reaction.

---

1Sponsored by the Air Force Office of Scientific Research

1We gratefully acknowledge support from The Dow Chemical Company under the University Project Initiative.
2:37PM L06.00005 Experimental investigation of disparate viscosity turbulent mixing in a coaxial jet mixer\textsuperscript{1}, MICHAEL AHMAD, GOKUL PATHIKONDA, MUSTAFA USTA, Georgia Institute of Technology, IRFAN KHAN, Dow Company, CYRUS AIDUN, DEVESH RANJAN, Georgia Institute of Technology — Industrial chemical processes often involve continuous mixing of streams of reactants at different viscosities. In contrast to streams of constant viscosity, the variable viscosity streams provoke different mechanisms for turbulence and mixing, which are known to affect chemical yield. These mechanisms need thorough study to enable predictive modeling of such phenomena. Thus, simultaneous PIV and PLIF are employed to measure the turbulent and mixing dynamics in a confined, co-axial jet flow with viscosity ratios up to 40. This data reveal the effects of the imposed viscosity gradients on the nature in which the jet spreads into the co-flow. Evidently, the viscosity disparity largely confines the vortical structures to the lower viscosity fluid resulting in larger and more intermittent scalar structures in the higher viscosity fluid. This overarching phenomena reflects itself within realizations of the mean velocity and scalar fields, but also the Reynolds stresses and turbulent and scalar dissipation. These observations allude to differences in expected chemical yield. This study is performed concurrently with a computational effort to develop insight for SGS models of turbulent mixing within mediums with viscosity gradients.

1We gratefully acknowledge support from The Dow Chemical Company under the University Project Initiative.

2:50PM L06.00006 Competitive-consecutive reaction of liquids with disparate viscosity\textsuperscript{1}, MUSTAFA USTA, GOKUL PATHIKONDA, MICHAEL AHMAD, Georgia Institute of Technology, IRFAN KHAN, Dow Company, DEVESH RANJAN, CYRUS AIDUN, Georgia Institute of Technology — Competitive-consecutive reactions between liquids with disparate viscosity come along with several challenges due to high Schmidt (Sc) number and high reaction rates. The relevant scales at high Sc for a scalar become much smaller than turbulent dissipative scales. Higher reaction rates turn the problem into mixing limited reactions. In this study, we employ large-eddy simulation (LES) to investigate the effect of viscosity ratio on the reaction yields. The focus is on the co-axial jet and jet in crossflow mixing. In addition to the computations, the experimental setup features PIV and PLIF measurements to resolve the velocity and the mixture fraction. The investigations not only the mixing and reaction characteristics are considerably affected by the viscosity ratios of the liquids. The results for the mixing will be presented with comparison to experiments for the viscosity ratio of up to 40. The chemical reaction results; and the challenges in the subgrid scale modeling of variable viscosity, high Schmidt number, and high reaction rates will be presented for various viscosity ratios.

1We gratefully acknowledge support from The Dow Chemical Company under the University Project Initiative

3:03PM L06.00007 Experimental Study of a Turbulent Liquid Flame. , MICHAEL LE BARS, CNRS, IRPHE UMR 7342, Marseille (France), BAPTISTE ISNARD, Ecole Normale Supérieure de Lyon et IRPHE UMR 7342, Marseille (France), CHRISTOPHE ALMARCHA, Aix-Marseille Univ, IRPHE UMR 7342, Marseille (France) — Turbulent deflagrations are of fundamental importance for industrial applications and security issues. When the environment is congested by numerous obstacles, the turbulence generated by the blast upstream of the front strongly influences the flame speed and its acceleration. Our objective is to characterize experimentally turbulent premixed combustion so as to improve its modeling and predictability. In particular, we aim at collecting experimental data for a critical analysis of flame velocity models, based on detailed non-intrusive measurements of local turbulence and front parameters. We investigate the coupling between turbulence and the premixed flame thanks to a so-called liquid flame, using an autocatalytic chemical reaction \[1\]: this chemical reaction generates an intermediate product that increases the reaction rate, leading to the building-up of a front with the final products in close similarity with premixed combustion. In our system, the turbulence is generated by 2 oscillating grids. PIV-PLIF measurements allow simultaneously quantifying the turbulence rate and the front evolution. The experimental results are used to challenge the predictions from classical theoretical and numerical models. [1] Shy et al. Combust. Flame 1999. Pocheau & Hariambet PFE 2006.

Monday, November 25, 2019 1:45PM - 3:16PM — Session L07 Non-Linear Dynamics and Chaos I

1:45PM L07.00001 A 3D View on Elastic Instabilities of Polymeric Solutions in Cross-Channel Flow, PAULO ARRATIA, BOYANG QIN, RANJINGSHANG RAN, University of Pennsylvania, PAUL SALIPANTE, STEVEN HUDSON, NIST — Over a decade ago Jerry Gollub along with two talented Haverford College undergraduate students and one of the authors (PEA) investigated the flow of viscoelastic fluids near hyperbolic points using a cross-slot channel. We found two novel flow instabilities of a planar extensional flow under steady forcing: (i) a symmetry breaking and steady instability and (ii) a time-dependent flow as strain-rate is increased. Here, new experiments using 3D particle tracking velocimetry are presented. We find a new symmetry-breaking instability normal to the extension plane, marked by a bi-stable flow switching in that direction and “buckling” of the separatrix between the impinging inflow streams. A strong connection exists therefore between temporal fluctuations and 3D flow. We explore this connection in some detail comparing the distribution of 3D modes with fluctuations of local pressure in inlet channels.

1:58PM L07.00002 Controlling thermal convection with sidewall heating, JUN ZHANG, MAC JINZI HUANG, New York University — Thermal convection of fluids takes place ubiquitously in nature and in our lives. It powers the motion of the earth’s mantle, drives winds in the atmosphere and currents in the oceans, and determines how fast we cook food and cool computers. Once a temperature difference is applied across a fluid, a thermal energy passes through the fluid from the heated bottom to the cooled top. This heat flow is known to depend on the applied temperature difference. Here we study a simple way to control or modify this heat flow by injecting an additional heat flow from the side of the fluid. Our thermal system is reminiscent of an electronic transistor where an electrical current controls another electrical current.

2:11PM L07.00003 Active matter and fluid dynamics from the cell, MICHAEL SHELLEY, Flatiron Institute (Simons Foundation) & Courant Institute (NYU) — Inside of cells, the nature of the forces that underlie important transport and developmental processes therein are often obscure. These forces are exerted through the activity of the cellular cytoskeleton, a collection of transitory biopolymers and their associated molecular motors. Fluid dynamics and fluid-structure interactions have a role to play in understanding the internal dynamics of cells; the fluidic cytoplasm both modulates and transmits forces generated by motion of the active cytoskeleton, while the induced flows can reveal how and where those forces are being produced. These same cytoskeletal components, when studied in vitro, outside of the cell, evince self-organizing dynamics reminiscent of the self-organization seen in mitotic spindles – the organelle that orchestrates the division of chromosomes – in early development.
2:24PM L07.00004 The Role of Disorder in Gravity-Driven Granular Flow . KERSTIN NORSTROM, GRACE CAI, ANNA BELLE HARADA, Mount Holyoke College — The gravity-driven flow of a granular material in a silo has been studied for many years due to its obvious practical importance. It has been shown that placing a fixed intruder near the exit aperture can suppress clogging and enhance flow, but the dominant mechanisms have been debated. We present results on the flow of monodisperse grains in a quasi-2D silo in the presence of a fixed intruder using high speed video to track the individual grain motions. We find the intruder dramatically affects not only the structural order of the packing, but also the spatiotemporal dynamics of the flow, creating flow that alternates but lacks a characteristic frequency or frequencies. We find a combination of geometrical effects and enhanced granular temperature are responsible for suppression of clogs in the presence of the intruder; the dominant effect depends on where the intruder is placed.

2:37PM L07.00005 Measurements of capillary-gravity wave reflection from surface intersecting structures. LIKUN ZHANG, XINYUE GONG, ROBERT LIRETTE, ZHEGUANG ZOU, The University of Mississippi — A full understanding of the fluid surface wave interactions with solid boundaries in micro-gravity environments is essential for the containment and control of liquids during manned space missions. When containing fluids using minimal support structures, capillary oscillations occur and capillary wave energy is dissipated at contact lines. We present results of experimental measurements on the reflection and transmission of capillary-gravity waves encountering a surface pieceing barrier. Dissipation near the contact lines for capillary waves is deduced from the transmission and reflection measurements. The surface perturbation is measured by the ultrasonic Doppler-shift method. We determine the dependence of the interactions with the wave frequency. The frequency is varied from the gravity wave limit to the short-wavelength regime where surface tension effects are dominant over gravity effects.

2:50PM L07.00006 Stretching Fields in Chaotic and Turbulent Fluid Flows . GREG VOTH, Wesleyan University — The stretching that fluid elements experience along their trajectories is responsible for many of the dominant features of passive scalar fields (such as an advected dye), passive director fields (such as advected fibers) and the vorticity field. Recent work on passive director fields has shown that fibers are usually aligned with their neighbors by fluid stretching. However, there are also thin walls across which the fiber orientation changes rapidly which we call alignment inversion walls. Experiments in a turbulent flow between oscillating grids and numerical simulations of homogeneous isotropic turbulence are used to explore the mechanism that produces alignment inversion walls in a fractal pattern in chaotic and turbulent flows. The orientation field of fibers provides not only fascinating mathematical structure but also a new way to study the dynamics of the turbulence.

3:03PM L07.00007 Dynamics of Carbon Nanotube Porins in Supported Lipid Bilayers . MARY LOWE, KYLIE SULLIVAN, JOSEPH LOPEZ, Loyola University Maryland, YULIANG ZHANG, ALEKSANDR NOY, Lawrence Livermore National Laboratory — Cell membranes in bacteria and eukaryotes possess transmembrane proteins called “porins,” which provide channels for ions and small molecules to enter or exit the cell. The cell membrane is composed of a phospholipid bilayer that behaves like a two-dimensional fluid in which lipids and membrane proteins laterally diffuse within the plane. Recently techniques have been developed to insert carbon nanotubes with a 1.5 nm diameter into a lipid bilayer to form “carbon nanotube porins” (CNTPs). Using high-speed atomic force microscopy, we recorded the movement of CNTPs in real time in a lipid bilayer supported on a mica surface. The diffusion coefficient was measured for varying concentrations of phospholipids DOPC and DMPC. The data reveal that the CNTP diffuses more quickly at lower concentrations of DMPC, exhibits fluctuations in tilt with respect to the membrane normal, and changes its azimuthal orientation within the membrane over time. Molecular dynamics simulations of a CNTP in DOPC:DMPC bilayers support our experimental findings and reveal additional information on molecular structures and movements of the CNTP and lipids. CNTPs can be a biomimetic platform for studying biological channels, or a means of developing stochastic sensors for measuring ionic transport through pores.

1Supported by DOE-VFP program, DOE contract DE-AC52-07NA27344, and NSF MRI grant 1626262.

Monday, November 25, 2019 1:45PM - 3:16PM — Session L08 Non Newtonian Fluid: Rheology 212 - Dana Grecov, University of British Columbia

1:45PM L08.00001 New methodology for rheological properties calculation of disotic nematic Liquid Crystals . DANA GRECOV, ARASH NIKZAD, University of British Columbia, ABOZAR AKBARI, Monash University — In this study, the main objective is to propose a methodology to calculate various rheological properties and viscosity coefficients for a disotic nematic LiquidCrystal (DNLC). Liquid Crystals (LCs) are anisotropic viscoelastic materials with fluid-like and crystal-like properties. The anisotropy of the viscosity coefficients, with respect to different flow directions, is a unique property of the liquid crystalline phase. Using the presented method, the required viscosity coefficients for different concentrations of Graphene Oxide (GO) dispersion, as a DNLC, were obtained. GO, the most processable derivative of graphene, is oxygen-functionalized graphene and has attracted enormous attention due to the unique liquid crystal and rheological properties. Shear-thinning rheological behavior of the nematic GO dispersion has opened an easy way to fabricate graphene-based devices in micro and macro scales. Our results showed that the alignment viscosity from the analytical method was consistent with the experimental one. Using the calculated viscosity coefficients, the numerical simulation results of the nematic GO under flow were consistent with the experimental results.

1The authors acknowledge financial support from the Natural Science and Engineering Research Council (NSERC) of Canada.

1:58PM L08.00002 Lubrication theory applied to the Landau deGennes theory . SOMESH BHATIA, DANA GRECOV, University of British Columbia — Liquid crystals (LCs) are anisotropic, viscoelastic materials with properties intermediate of solids and liquids. They are useful structural and functional materials; due to their ability to form ordered layers close to the bounding surfaces they are used as lubricants. The material properties and the behavior of LCs are dependent on the microstructure of the LCs which is affected by the strength of the hydrodynamic field and elastic effects of the bounding surfaces. Working towards the goal of studying thin films of LCs, the tensorial Landau de-Gennes theory is simplified using the Reynolds scaling approach. The solution of the fully coupled system of Navier-Stokes equations with a modified stress tensor which accounts for the viscoelastic contribution and the equations of Landau de-Gennes (LdG) theory was obtained using COMSOL Multiphysics™. The simplified equations derived from the lubrication theory were solved on MATLAB and the results were validated using a Couette flow by comparison with the simulations of the fully coupled system. As the LdG theory is a multiscale theory, the solution of the coupled system requires important computational resources. The advantage of the simplified equations of lubrication theory is that it allows the reduction of study time.

1The financial support of the Natural Sciences and Engineering Research Council (NSERC) of Canada is gratefully acknowledged.
2:11PM L08.00003 Electrostatics and Rheology in Semidilute Polyelectrolyte Solutions, GUANG CHEN, ANTONIO PERAZZO, HOWARD STONE, Princeton University — Polyelectrolyte (PE) solutions, which are charged polymers in aqueous solvents, have been studied from many perspectives in the past century. However, the effect of added salt on semidilute PE solutions remains unclear. To understand the electrostatic interactions among the polymers and electrolyte ions, we use a mean-field approach to determine the electrostatic energy of a PE solution at various polymer concentration $n_p$ and salt concentration $n_s$. We derive asymptotic approximations for the potential and ion distributions in nano-confined charged systems by probing the Poisson-Boltzmann equation within and beyond the Debye-Hückel linearization, and obtain distinct scaling laws for the electrostatic energy and viscosity for PE solutions at different regimes of $n_p/n_s$. One of our predictions coincides with the empirical Fuoss law for viscosity $\eta \propto n_p^{1.5}$ under the same assumption $n_s \ll n_p$, as in de Gennes scaling theory. Our theory also captures an unexplained empirical observation $\eta \propto n_p^{0.68}$ for salt-free PE solutions [1], and provides more physical insights on the effects of salt and charge fraction on the properties of PE solutions. [1] C. G. Lopez, R. H. Colby, P. Graham and J. T. Cabral, Macromolecules, 50, 332 (2017)

2:24PM L08.00004 Influence of pH value on gel reaction for fluid flow pattern in a circular flow pipe, MASAKI YAMAGUCHI, Graduate school of science and technology, Keio University, TAKESHI YOKOMORI, TOSHIHISA UEDA, Department of mechanical engineering, Keio University — Influence of pH value for flow pattern and pressure variation with PVA–borax gel reaction in a circular flow pipe are experimentally investigated. The working fluids are 10 mass% polyvinyl alcohol (PVA) solution and 3 mass% borax solution. The pH value of borax solution is varied from 6.9 to 11.7 using sodium hydroxide (NaOH) or hydrogen chloride (HCl) solution. Three distinct flow patterns with gel reaction in a flow pipe are described at each pH value as, 1. Parallel flow, 2. Capsule flow and 3. Clogging. In the parallel flow, the injected borax is stretched by main flow velocity and it smoothly flows to downstream direction. The pressure is not varied since the gel reaction does not effect for flow pattern. In the capsule flow, the injected borax forms capsule shape and it flows to downstream region. The capsule is produced by gel sheet which is generated the interface between the injected borax and PVA solution. As a result, the injected borax is separated from PVA region. In clogging, injected borax forms fine finger shape product. Finally, injected material stays on the main flow pipe. The pressure cannot recover to the initial pressure because formed gel is adhered to the wall of main flow pipe. Thus, the pressure monotonically increases with time.

2:37PM L08.00005 Macromolecular Architecture and Complex Viscosity, ALAN JEFFREY GIACOMIN, MONA KANSO, CHAIMONGKOL SAENGOW, JOURDAIN PIETTE, Queen’s University — General rigid bead-rod theory [Hassager, J Chem Phys, 60, 4001 (1974)] explains polymer viscoelasticity from macromolecular orientation. By means of general rigid bead-rod theory, we relate the complex viscosity of polymeric liquids to the architecture of axisymmetric macromolecules. In this work, we explore the zero-shear and complex viscosities of 24 different axisymmetric polymer configurations. When non-dimensionalized with the zero-shear viscosity, the complex viscosity depends on the dimensionless frequency and the sole dimensionless architectural parameter, the macromolecular lopsidedness. In this work, in this way, we compare and contrast the elastic and viscous components of the complex viscosities of macromolecular chains that are straight, branched, ringed, or star-branched. We explore the effects of branch position along a straight chain, branched-chain backbone length, branched-chain branch-functionality, branch spacing along a straight chain (including pom-poms), the number of branches along a straight chain, ringed polymer perimeter, branch-functionality in planar stars, and branch dimensionality.

2:50PM L08.00006 Rheology of Pluronics: morphological transitions induced by temperature and concentration, ROSSANA PASQUINO, ALFONSO DI SARNO, MARINA D’APUZZO, SALVATORE COSTANZO, DImPai, Universit degli Studi di Napoli Federico II — Pluronics are a class of water-soluble triblock copolymers made by a sequence polyethylene oxide (PEO)-polypropylene oxide (PPO)-polyethylene oxide (PEO) segments. Due to the presence of hydrophilic and hydrophobic parts on the same molecule, they have the ability to self-assemble in water. By varying the molecular weights of EO and PO sections, it is possible to obtain different types of Pluronics, recognized by a different code. In this work, we study the structures detected in aqueous solutions of Pluronic F68 at different temperatures by means of Small Angle X-ray Scattering (SAXS) and Small Angle Neutron Scattering. Various concentrations ranging between 10% and 80% by weight of Pluronic in water were tested. We performed dynamic temperature ramp tests in linear regime at different ramp rates and we were able to evaluate, via a rheological extraction, the temperatures at which transitions occur. The temperature at which the transitions appear were measured as a function of the triblock-copolymer concentration. Molecular structures were analyzed via SAXS measurements, which showed a liquid to body-centered cubic phase transition. The results allowed for the determination of a complete rheological phase diagram water/F68.

3:03PM L08.00007 Dynamics of DNA-bridged particle dimers in well-entangled polymer solutions under large amplitude oscillatory shear (LAOS), SEUNGHWAN SHIN, KEVIN DORFMAN, XIANG CHENG, Department of Chemical Engineering and Materials Science, University of Minnesota, Minneapolis, MN 55455 — Although evidence for shear-banding flows in highly entangled polymer solutions has accumulated over the last two decades, the shear-induced microscopic conformational changes of individual chains that trigger shear banding remains unknown. Here, using a custom-built high-resolution rheo-confocal shear cell, we experimentally study the dynamics of DNA-bridged particle dimers in the shear-banding flow of well-entangled double-stranded DNA (dsDNA) solutions under LAOS to reveal the microscopic dynamics of entangled DNA chains in shear-banding flows. In our experiments, we first confirm that the velocity profiles of the entangled DNA solutions are inhomogeneous at high Weissenberg number (Wi) and develop into strong shear-banding flows with two distinct shear bands. Second, we investigate the dynamics of the particle dimers linked by long linear dsDNA chains and measure the distribution of dimer orientations in the high and low shear-rate bands. Quantitative analyses of the spatially distinct dynamics of such DNA-bridged dimers in the two co-existing bands provide important insights into the microscopic origin of the shear-banding flows in entangled polymer solutions.

1This research is supported by NSF CBET-1700771.
1:45PM L09.00001 Control of the Aerodynamic Loads on a 3-D Wing using Distributed Active Bleed. M.E. DESALVO, D. HEATHCOTE, M. SMITH, A. GLEZER, Georgia Institute of Technology — The aerodynamic loads on a 3-D wing with a trailing-edge flap are controlled in wind tunnel experiments using active distributed bleed of ambient air that is driven through arrays of surface openings by the inherent pressure differences between the pressure and suction surfaces and regulated by integrated lateral louvers. Interaction between the bleed and the local cross flow over the surface induces large-scale changes in the global flow field which lead to direct modification of the aerodynamic loads and thereby can augment conventional electromechanical control surfaces without changing angle of attack or flap deflection. The leading edge actuator (LEA) is designed to take advantage of the inherent temporal modulation of the bleed through different spanwise segments of the wing over a range of angles of attack and flap deflections. Stereo PIV measurements of the wake acquired phase-locked to the actuation in a streamwise-normal plane downstream of the model show the temporal effects of bleed on distributions of streamwise vorticity (including the tip vortex) and on the spanwise loading of the wing.

1Supported by ONR.

1:58PM L09.00002 A numerical study on the delay of airfoil stall by traveling wave surface morphing. AMIR AKBARZADEH, IMAN BORAZJANI, J. Mike Walker ’66 Department of Mechanical engineering, Texas A&M University, College Station, TX, 77840 — Surface morphing is found to be an energy efficient active flow control technique. While these morphings are typically, in the form of a simple vibration or a standing wave, a traveling wave morphing might to be more effective because traveling wave oscillations are thought to reattach the flow over aquatic swimmers and bluff bodies. Large eddy simulations (LES) of flow over a low Reynolds number (Re=50,000) airfoil at stall angle, which has a oscillatory suction side, are performed to investigate the effect of the morphing function, i.e., backward traveling waves, standing waves, and forward traveling waves, on reducing flow separation. In addition, the effects of frequency and amplitude of the wave are investigated. It has been observed that backward traveling waves can be more effective than other types of oscillations, including forward traveling waves and standing waves.

1This work was partly supported by the National Science Foundation (NSF) CAREER Grant CBET 1453982, and the High Performance and Research Center (HPRC) of Texas A&M University.

2:11PM L09.00003 Characterizing Prescribed Surface Morphing for Aerodynamic Flows: Mechanistic Insights for Control. KEVIN TRINER, ANDRES GOZA, University of Illinois at Urbana-Champaign — Robust and efficient flow control is key to improving the maneuverability and disturbance rejection of micro- and unmanned aerial vehicles. Previous and current actuation strategies to achieve this aim include zero-net mass flux (synthetic), plasma, and combustion actuators. We investigate an alternative actuation framework in which prescribed deformations are imposed along the suction surface of the airfoil. This prescribed surface morphing, which is a variant of the actuation methodology suggested by Jones et al. (2016), is designed to take the form of a traveling wave. This actuation strategy is promising because it has the ability to affect flow over the entire suction surface of the airfoil, and is not limited to specific streamwise locations. We use high-fidelity nonlinear simulations to quantify the effect of the surface morphing function, i.e., backward traveling waves, standing waves, and forward traveling waves, on reducing flow separation. In addition, the effect of frequencies and amplitude of the wave are investigated. It has been observed that backward traveling waves can be more effective than other types of oscillations, including forward traveling waves and standing waves.

2:24PM L09.00004 Transitory, Bi-Directional Control of the Aerodynamic Loads on an Airfoil in an Attached Flow. YUEHAN TAN, A. GLEZER, Georgia Institute of Technology, R. PATTERSON, P. FRIEDMANN, University of Michigan — The aerodynamic loads on a VR-12 airfoil are regulated bi-directionally to enable mitigation of structural vibrations when the base flow is fully-attached by using pulsed fluidic control upstream of the trailing edge. Actuation is effected using independently controlled high aspect ratio bi-stable, fluidically-switched actuation jets on the pressure and suction surfaces at nominally 0.88c. The transitory actuation temporally manipulates the circulation around the airfoil and its Kutta condition yielding changes in the aerodynamic loads. For example, it is shown that time-invariant actuation on the pressure and suction surfaces can lead to respective lift increments CL of up to +0.73 and -0.65 with relatively low drag penalty. The transitory characteristics of the aerodynamic loads to pulsed actuation are investigated with specific emphasis on the initial response following the onset of the actuation along with corresponding unsteady vortex shedding near the trailing edge using phase-locked particle image velocimetry. Supported by Vertical Lift Research Center of Excellence (VLRCOE).

2:37PM L09.00005 ABSTRACT WITHDRAWN —

2:50PM L09.00006 Manipulation of Streamwise Vortices by Air Injection on Trapezoidal Vane-Type Vortex Generators. GIOVANNI NINO, University of Washington, LUCAS WEBER, Technical University of Berlin, ROBERT BREIDENTHAL, University of Washington — This work investigated the feasibility of manipulating a streamwise vortex position by injecting pressurized air through the leading edge of a trapezoidal vortex generator (VG). The VGs were tested on a miniature wind tunnel. A pressure sensor was mounted on a two-dimensional traverse to gather dynamic pressure cross sections downstream of the vortex generator. From these pressure profiles, the vortex position was measured to determine the influence of the air injection. Supplied air pressure was varied as well as the location of the injection at three different positions along the leading edge. In addition, vortex generators with small round holes as injection ports as well as rectangular slots were investigated. Experimental results showed that injection at different VG leading-edge positions were able to displace the vortex in a controllable way.

3:03PM L09.00007 Passive drag reduction on a sphere using polyhedral designs. NIKOLAOS BERALTIS, KYLE SQUIRES, School for Engineering of Matter, Transport and Energy, Arizona State University, ELIAS BALARAS, Department of Mechanical and Aerospace Eng., George Washington University — It is established today that dimples are efficient in accelerating the drag crisis on a sphere reducing the drag coefficient at much lower Reynolds numbers when compared to a smooth sphere. Recently we reported Direct Numerical Simulations (DNS) demonstrating that the large difference in the minimum drag coefficient between a dimpled and smooth sphere in the post-critical regime comes from the dimples themselves as the flow separates and reattaches inside them. In this talk we propose a new class of geometries based on polyhedral designs resembling a faceted sphere. Wind tunnel testing and DNS demonstrate drag reduction by as much as 15% compared to a typical dimpled sphere, without affecting the critical Reynolds number marking transition to the post-critical regime. We utilize the experiments to establish the behavior of the drag coefficient as a function of the Reynolds number for different geometries and then use DNS for selected cases to obtain a detailed understanding of the flow physics. We will show important differences in the evolution of the boundary layers between a polyhedral and dimpled sphere as well as their wakes that reveal the underlying mechanism for the reduction in the drag coefficient.
Research under Award Number DE-SC0019290.

interface between any two consecutive partitions. We test our framework using different convection-dominated, unsteady-flow problems.

we introduce a long short-term memory (LSTM) neural network architecture along with a principal interval decomposition (PID) framework as

global nature often causes a deformation of the generated bases, especially in convective flows. Based on dimensionality reduction using POD,

framework. Proper orthogonal decomposition (POD) is well-known for its optimality in representing complex systems. However, its intrinsic

mathematical form of the driving physics. Conventional projection-based reduced order modeling (ROM) techniques can satisfy the first two

stationary conditions for the time-continuous minimization problems are obtained by deriving the associated Euler-Lagrange equations. Both
direct (discretize then minimize) and indirect (minimize and then discretize) solution techniques are explored. The proposed approach displays

commonalities with optimal control problems and can be viewed as a generalization of the popular Least-Squares Petrov-Galerkin (LSPG)
method. By formulating the residual minimization problem from the time-continuous level, the TC-LSRM approach overcomes the time-step
sensitivity and time-scheme dependence that LSPG is subject to. Numerical experiments demonstrate that the proposed approach can lead to
more accurate and physically relevant solutions than existing model reduction approaches.

This approach significantly outperforms linear methods, such as POD, for drastic dimensionality reduction. Furthermore, evolving the nonlinear
spatiotemporal dynamics in translation-symmetric systems via deep learning

is presented. The proposed approach, referred to as Time-Continuous Least-Squares Residual Minimization (TC-LSRM), sequentially minimizes the time-continuous full-order model residual within a low-dimensional trial space over a series of time slabs. The stationary conditions for the time-continuous minimization problems are obtained by deriving the associated Euler-Lagrange equations. Both
direct (discretize then minimize) and indirect (minimize and then discretize) solution techniques are explored. The proposed approach displays

commonalities with optimal control problems and can be viewed as a generalization of the popular Least-Squares Petrov-Galerkin (LSPG)
method. By formulating the residual minimization problem from the time-continuous level, the TC-LSRM approach overcomes the time-step
sensitivity and time-scheme dependence that LSPG is subject to. Numerical experiments demonstrate that the proposed approach can lead to
more accurate and physically relevant solutions than existing model reduction approaches.

This work is supported by ONR grant N00014-18-1-2865 and AFOSR grant FA9550-18-1-0174.

1Australian Research Council grant DP140104402

1This is a material based upon work supported by the U.S. Department of Energy, Office of Science, Office of Advanced Scientific Computing Research under Award Number DE-SC0019290.

2:24PM L10.00004 Nonlinear dimensionality reduction and prediction of chaotic spatiotemporal dynamics in translation-symmetric systems via deep learning

2:11PM L10.00003 LSTM based nonintrusive ROM of convective flows

1:58PM L10.00002 Model Reduction via Time-Continuous Least-Squares Residual Minimization

1:45PM L10.00001 A Bayesian Reinterpretation of Dynamical System Identification by Sparse Regression Methods

1:45PM L10.00001 A Bayesian Reinterpretation of Dynamical System Identification by Sparse Regression Methods1, ROBERT K NIVEN, The University of New South Wales, Australia, LAURENT CORDIER, Institut Pprime, Poitiers, France, MARKUS ABEL, MARKUS QUADE, Ambrosys GmbH, Potsdam, Germany, ALI MOHAMMAD-DJAFARI, CentraleSupélec, Gif-sur-Yvette, France. — Recently, many researchers have developed sparse regression methods for the identification of a
dynamical system from its time-series data. We demonstrate that these methods fall within the framework of Bayesian inverse methods. Indeed,
the Bayesian maximum a posteriori method, using Gaussian likelihood and prior functions, is equivalent to Tikhonov regularization based on
Euclidean norms. This insight provides a Bayesian rationale for the choice of residual and regularisation terms for any problem, respectively
from the Bayesian likelihood and prior distributions. It also provides access to the full Bayesian inversion apparatus, including estimation of
uncertainties in the inferred parameters and the model, explicit calculation of the optimal regularisation parameter, and the ranking of competing
models using Bayes factors. In addition, advanced Bayesian methods are available to explore the inferred probability distribution of the model,
should this be desired. We demonstrate these points by analysis of several dynamical systems using standard and Bayesian sparse regression
methods. We also discuss the estimation of intermediate parameters and their handling within a Bayesian framework.

1This is a material based upon work supported by the U.S. Department of Energy, Office of Science, Office of Advanced Scientific Computing Research under Award Number DE-SC0019290.

3A - Nathan Kutz, University of Washington
2:50PM L10.00006 A Koopman-based framework for forecasting the spatiotemporal evolution of chaotic dynamics. MOHAMMAD AMIN KHODKAR, PEDRAM HASSANZADEH, ATHANASIOS ANTOULAS, Rice University — We show the remarkable skills of a data-driven method in spatiotemporal prediction of high-dimensional and chaotic dynamics. The method is based on a finite-dimensional approximation of Koopman operator where the observables are vector-valued and delay-embedded, and the nonlinearities are treated as external forcings. The predictive capabilities of the method are demonstrated for well-known prototypes of chaos such as the Kuramoto-Sivashinsky equation and Lorenz-96 system, for which the data-driven predictions are accurate for several Lyapunov timescales. Similar performance is seen for two-dimensional lid-driven cavity flows at high Reynolds numbers.

3:03PM L10.00007 Predicting long-term dynamics of chaotic systems with hybrid machine learning1, SREE TEJ LAKKAM, BALAMURALI B T, Singapore University of Technology and Design, JURRIAAN J J GILLISSEN, Department of Mathematics, University College London., ROLAND BOUFFANAI S, Singapore University of Technology and Design — Forecasting of chaotic systems relies on estimating long-term dynamics of the system to make reasonable predictions. Our work aims to use an efficient hybrid machine learning technique to improve the estimation of long-term statistics while being resilient to short-term anomalies in determining future states of the system. Our hybrid machine learning technique combines a Long Short Term Memory (LSTM) architecture and ensemble modeling. LSTM is used to extract the long-term dependencies in the chaotic system data, while ensembling perturbs and combines multiple LSTMs to obtain better predictive performance compared to any of the constituent LSTM alone. We demonstrate the forecasting capability of this framework using time-series data from a Lorenz system and subsequently apply it to planar homogeneous turbulence flow field. Using visual verification and power spectra analysis, we conclude that our model can learn and predict the long-term dynamics even when short-term forecasting fails owing to the inherent unpredictability of chaotic systems. These results have far-reaching implications for the use of machine learning in fluid mechanics. Moreover, this approach is completely data-driven and relies on the LSTM capability to capture the long-term statistics of a chaotic system. 1 MOE-SUTD PhD fellowship and SUTD grants (IDC: IDG31800101 and AI Sector: PIE-SGP-AI-2018-01)

Monday, November 25, 2019 1:45PM - 3:16PM — Session L11 Experimental Techniques: Microscale 3B - Ralph Budwig, U. Idaho

1:45PM L11.00001 Microfluidic Study of Temperature Dependent Phase Changes in Aqueous Aerosol Systems1, PRIYATANU ROY, CARI DUTCHER, University of Minnesota — Atmospheric aerosols are suspensions of small liquid or solid particles suspended in air which play a major role in climate change, either directly by absorption and scattering of solar irradiation or indirectly by cloud formation. They are complex multiphase fluid systems and experience a broad range of relative humidities and subzero temperatures in the atmosphere affecting their phase state i.e. well-mixed liquid, phase separated or crystallized solid. In situ observation of relative humidity and temperature of aerosol droplets are difficult and expensive. In this work, low cost microfluidic devices are used to generate and study phase states of aqueous droplets of real aerosols and chemical mimics in static and flow-through devices at subzero temperatures. The static device dehydrates trapped droplets in quasi-equilibrium to study the effect of relative humidity and water loss on phase state at different temperatures. The high-throughput flow-through microfluidic device measures ice nucleation temperature of aerosols. The results will be used to predict the cloud and ice formation activity of atmospheric aerosols. 1 NSF CAICE and UMN MNC

1:58PM L11.00002 Droplet Nucleation and Condensation on a Hydrophilic Surface. SHAHAB BAYANI AHANGAR, JEFFREY ALLEN, Michigan Technological University, SEONG HYUK LEE, Chung-Ang University, CHANG KYOUNG CHOI, Michigan Technological University — Dropwise condensation is a ubiquitous phenomenon in nature. Dropwise condensation has the potential to improve the efficiency of condensing surfaces and reduce the maintenance costs of systems. However, efforts to design and fabricate surfaces that can sustain long-term dropwise condensation have not been successful. The main reason is that the nucleation physics, which are key to understanding degradation, behind dropwise condensation are not fully understood; thus, researchers have mostly relied on a trial and error approach for developing new surfaces. In this work, a series of fundamental experiments were done to identify the governing mechanism of dropwise condensation on smooth hydrophilic surfaces by probing the solid-vapor interface during phase-change. The results evaluate the existence and structure of the thin film and initial nuclei that develop during condensation. The adsorption kinetics theory is used to improve understanding of the dropwise condensation mechanism during droplet formation at the onset of condensation.

2:11PM L11.00003 Measurement of biofilm stresses in laminar flows by a digital holography interferometry (DHI) and an embedded wrinkle free thin-film polymer mirror. MARYAM JALALI, JIAN SHENG, Texas A&M — Bacteria are unicellular microorganisms that commonly exist in either planktonic and biofilm lifestyles. Biofilms have viscoelastic nature and are subject to deformation under external disturbances (e.g. fluid shear stress). Here, we developed a technique to perform in-situ measurement of viscous stresses exerted biofilm under different flow velocities. The experiments are performed in a uniquely developed microfluidic platform composed of a flexible thin-film mirror embedded in Polydimethylsiloxane (PDMS), that performs as a stress sensitive substrate. This 30nm aluminum thin film is sandwiched between two PDMS layers, and is free of wrinkles and cracks in addition to being specular reflective. The microfluidics is attached to a chemostat and two peristaltic pumps that continuously flow bacterial suspensions in close-loop to facilitate biofilm growth and generate flow shear. DHI measures nano-strain of the thin-film and consequently stresses via finite element modeling of thin film.
2:24PM L11.00004 Off-axis digital holographic microscopy (DHM) as a method of high temporal and spatial resolution flow visualization¹. MANUEL BEDROSSIAN, Department of Medical Engineering, California Institute of Technology, 1200 E. California Blvd. Pasadena CA 91125, KURT LIEWER, CHRIS LINDENSMITH, Jet Propulsion Laboratory, California Institute of Technology. 4800 Oak Grove Dr. Pasadena CA. 91011, JPL-NEXT TEAM — Off-axis digital holographic microscopy is a developing optical modality that is capable of achieving diffraction limited resolution on the sub-micron spatial scale at high temporal resolution and across a large depth of field. By using two coherent beams of light that become recombined at the optical detector, off-axis of each other, the 3D optical information of the sample becomes encoded in the resulting interference pattern (hologram). Numerical processing of the hologram allows the volumetric reconstruction of the sample. The temporal resolution is limited by the frame rate of the camera used to record the holograms and thus dynamic processes such as fluid flows can be imaged and visualized by seeding the flow with appropriately sized particles. Our current implementation of an off-axis DHM is capable of diffraction limited resolution of 780 nm across an interrogation volume of 365x365x900 cubic microns. Hardware and software developments in off-axis DHM show this optical modality to be a promising method of high throughput flow visualization on the microscale.

²Funding for this project was provided by the JPL-NEXT program.

2:37PM L11.00005 Structured-illumination microscopy to improve the spatial resolution of microscale particle velocimetry². MICHAEL SPADARO, MINAMI YODA, Georgia Institute of Technology — The spatial resolution of optical methods in microchannel flows such as microscale particle image velocimetry (μPIV) is often limited because the entire flow volume is illuminated, and signal from tracers beyond the focal plane affects the measurement. Structured-illumination microscopy (SIM), originally developed for optical sectioning of stationary objects (e.g. cells) in fluorescence microscopy, is a promising way to acquire planar slices of the flow with a thickness comparable to the depth of field of the imaging system. SIM reconstructs the signal from the focal plane using multiple “raw” images of the flow illuminated by a sinusoidally varying (i.e., structured) intensity distribution at different phases using (spatial) frequency mixing. Initial results are presented here for microscale particle velocimetry using SIM images reconstructed from two raw images of laminar Poiseuille flow seeded with fluorescent polystyrene microparticles through a microchannel. The velocities obtained using SIM-based particle velocimetry are compared with results obtained from “standard” μPIV obtained with volume illumination in the same flow, and used to estimate the spatial resolution of this new technique.

²Supported by US Army Research Office

2:50PM L11.00006 A Novel Transparent Sediment Simulant for Unveiling the Bed Topography and Interstitial Processes, BRANDON HILLIARD, RALPH BUDWIG, DANIELE TONINA, JEFF REEDER, University of Idaho, RICHARD SKIFTON, Idaho National Laboratory — Particle Image Velocimetry (PIV) and Planar Laser-Induced Fluorescence (PLIF) are two commonly used and powerful laboratory experimental methods for whole-field velocity measurements and flow visualization. When they are coupled with refractive index matching (RIM), they can map both velocity fields and porous media architecture. We present a study that utilizes RIM coupled-PIV and RIM coupled-PLIF methods to not only quantify the flow within a packed bed of irregular shaped grains but also to map the internal structure of the porous media. We use a fluoro-carbon polymer with a specific gravity of 1.97 and optical properties similar to that of water as a simulate for sediment grains within a flow cell. These irregular grains, varying in shape and size, provide a structure that may simulate stream bed sediment and other porous media of granular materials. Here, we present the first experiments and discuss the image processing of both PIV and PLIF experiments, the image quality necessary for mapping the grain bed, and its errors and limitations. Our results pave the way for a novel application where both biological (microbial growth) and physical processes can be studied simultaneously.

3:03PM L11.00007 Added mass of porous solids oscillating in dense gas³. XIAOLONG YIN, SIRADON PRATEEPSWANGWONG, KEERTHANA KRISHNAN, Colorado School of Mines — Oscillation frequency of an object submerged in fluid is usually detectably retarded by the mass of co-accelerated fluid. In this study, this added-mass effect is explored for porous solids oscillating in dense gas, using an enclosed spring-mass system at high pressures. Compared with a non-porous solid of the same shape, the added mass of a porous solid is always higher because of pore-residing gas. When the period of oscillation is much greater than the time of viscous relaxation inside the pores, pore-residing gas follows the motion of the porous solid, and its mass can be estimated using the difference in the added mass between porous and non-porous solids. Experiments conducted using Berea sandstone show that masses of pore-residing gas obtained from oscillations were in good agreement with those calculated using pore volume of the sandstone and gas densities. Added mass of porous solids observed from oscillations can therefore serve to give pore volume when gas density is known, or gas density when pore volume is known. Additional added mass was noticed, however, when mesoporous solids (nanoporous silica and rocks) were used with a condensable gas.

³Mines Undergraduate Research Fellowship, Chevron International Fellowship

Monday, November 25, 2019 1:45PM - 3:16PM — Session L12 Vortex Dynamics and Vortex Flow: Instability 303 - Eckart Meiburg, UCSB

1:45PM L12.00001 Turbulence generation through an iterative cascade of the elliptical instability¹. RYAN MCKEOWN, Harvard University, RODOLFO OSTILLA-MONICO, Houston University, ALAIN PUMIR, ENS Lyon, MICHAEL BRENNER, SHMUEL RUBINSTEIN, Harvard University — We demonstrate the existence of a novel mechanism in which two counter-rotating vortices violently collide and break down, leading to the rapid development of a turbulent energy cascade mediated by iterations of the elliptical instability. We probe the full 3D dynamics of this breakdown by conducting both experimental visualizations of colliding vortex rings and numerical simulations of colliding vortex rings and vortex tubes. We observe how the onset of the elliptical instability causes the vortex cores to develop antisymmetric perturbations, which give rise to an ordered array of secondary vortex filaments, perpendicular to the original cores. Adjacent secondary filaments counter-rotate and interact with each other in the same manner as the original configuration. In the high-Reynolds number limit, we observe after a few iterations of this instability, whereby a new generation of tertiary filaments forms perpendicular to the interacting secondary filaments. The energy spectrum of this turbulent breakdown exhibits ~ k⁻⁵/³ scaling, a hallmark of homogeneous isotropic turbulence. We find that the elliptical instability must play a major role in the formation and sustenance of turbulent flows by providing a means through which energy is conveyed down to dissipative scales.

¹National Science Foundation: DMR-1420570, DMS-1411694, and DMS-1715477.
1:58PM L12.00002 Vortical structures of an axisymmetrically oscillating self-excited jet subjected to transverse acoustic forcing. ABHIJIT KUMAR KUSHWAHA, The Hong Kong University of Science and Technology, NICHOLAS WORTH, JAMES R. DAVISON, Norwegian University of Science and Technology, MAE-HKUST/EPT-NTNU COLLABORATION COLLABORATION — We experimentally examine the vortical structures in the near-field region of an axisymmetrically oscillating self-excited low-density jet subjected to axial and transverse acoustic forcing. We apply the forcing at frequencies around the global frequency of the jet and measure its response via time-resolved stereoscopic particle image velocimetry. We find that, when forced at amplitudes sufficient for synchronization, the jet exhibits two distinct roll-up of shear layers when the forcing is axi and (ii) an inverse roll-up of shear layers when the forcing is transverse. We find that the latter type coincides with a suppression of the self-excited global mode via asynchronous quenching. Using proper orthogonal decomposition, we extract the dominant modes of the jet, resolving the flow structures associated with acoustic and vortical disturbances. As well as providing new insight into the way external acoustic oscillations interact with self-excited hydrodynamic oscillations, this study clarifies the role of symmetry breaking in suppressing global instability in axisymmetrically oscillating self-excited jets.

2:11PM L12.00003 Topology and dissipation during vortex reconnection. ROBERT M. KERR, University of Warwick — The evolution of the vortical, topological and geometric properties of several configurations of vortices are compared. Trefoil vortex knots, coiled rings, anti-parallel perturbed vortices and colliding rings and one goal is to identify what the influence twist, writhe, linking numbers and helicities have upon the generation, or suppression, of energy dissipation. The focus will be upon the latest anti-parallel vortices reconnection calculations for which the growth of the enstrophy $Z$, the volume-integrated vorticity squared, is consistent with $\nu$-independent energy dissipation $\Delta \epsilon = \int_0^t \epsilon \ dt$, $\epsilon = \nu \Delta Z$ when the Reynolds number is high enough and the domain is large enough. This is consistent with the $\nu$-independent growth of $\epsilon$ for trefoil vortices (JFM 839, R2, 2018) and the type of reconnection structures observed when vortex rings collide (McKeown et al. PR Fluids 3, 124702, 2018). However, the $Z \sim (T/\nu)(r-v)^{-2}$ (or linear $B_0(t) = (\nu T/2)^{-1/2}$) regime seen for trefoils and nested coiled rings (JFM 854, R2, 2018) is not observed. Could the geometric numbers and helicity be why? The helicity of anti-parallel vortices is identically zero, while the others' topological helicity = twist + writhe is large.

2:24PM L12.00004 Flow-Induced Flapping Foil Instability. JIAQI MAI, PAUL F. FISCHER, ARNE J. PEARLSTEIN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — We have conducted a computational investigation of two-dimensional self-excited flapping-foil instability in the channel formed by two parallel plates, up to Reynolds numbers (based on average velocity and plate separation) of 400, and the effects on heat transfer. Our approach, for a massless foil, spatially discretizes the incompressible Navier-Stokes equations by a spectral-element method, while the dynamical equation describing foil deformation is discretized by a p-type finite element method. The spectral-element mesh for fluid temporally moves, based on an arbitrary Lagrangian-Eulerian (ALE) formulation, so that zero-thickness foil is always attached to the mesh and the no-slip condition is satisfied. Nonlinear terms in the incompressible Navier-Stokes equations are advanced explicitly in time, while linear terms are advanced implicitly. The structural equations are advanced in a weakly coupled sense, consistent with semi-implicit temporal discretization of Navier-Stokes, and the ALE moving-mesh strategy. We update the flow from Navier-Stokes, with a prescribed zero relative velocity on the foil, by superposing a series of Stokes solutions, each representing unit acceleration for one discretized degree of freedom of the foil, and weighted to satisfy the structural equation. This approach efficiently avoids the numerical instability of traditional partitioned fluid-structure interaction algorithms in the zero-mass case.

2:37PM L12.00005 Coupling of vortex breakdown and stability in a vortex T-mixer flow. SAN TO CHAN, Okinawa Institute of Science and Technology Graduate University, JESSE AULT, Biomedical Sciences, Engineering, and Computing Group, Oak Ridge National Laboratory, SIMON HAWARD, Okinawa Institute of Science and Technology Graduate University, ECKARTE MEIBURG, Department of Mechanical Engineering, University of California, Santa Barbara, AMY SHEN, Okinawa Institute of Science and Technology Graduate University — We use microfluidic experiments and numerical simulations to study the flow in a vortex T-mixer: a T-shaped channel with staggered, offset inlets. The vortex T-mixer flow is characterized by a single dominant vortex, the stability of which is restabilized by the vortex. Finally, a third vortex breakdown region appears at $Re \sim 90$, a first vortex breakdown regime seen for trefoils and nested coiled rings (JFM 839, R2, 2018) is not observed. Could the geometric numbers and helicity be why? The helicity of anti-parallel vortices is identically zero, while the others' topological helicity = twist + writhe is large.

2:50PM L12.00006 Influence of bottom topography on vortex stability. BOWEN ZHAO, Yale University, EMMA CHIEUSSE-GERARD, ENSTA ParisTech, GLENN FLIERL, MIT — The effects of topography on the linear stability of both barotropic vortices and two-layer, baroclinic vortices are examined by considering cylindrical topography and vortices with stepwise relative vorticity profiles in the quasi-geostrophic approximation. Four vortex configurations are considered, classified by the number of relative vorticity steps in the horizontal and the number of layers in the vertical. In the barotropic calculation, the vortex is destabilized by topography having an oppositely signed potential vorticity jump while stabilized by topography steps in the horizontal and number of layers in the vertical. This is consistent with the barotropic calculation except that the growth rate weakens and, for a two-step vortex, becomes less sensitive to topography (sign and magnitude) as baroclinicity increases. The smaller growth rate for a baroclinic vortex is consistent with previous findings that vortices with sufficient baroclinic structure could cross the topography relatively easily.

3:03PM L12.00007 Analysis of the transition between Kelvin’s equilibria using proper orthogonal decomposition. MIRA KIM, Concordia University, HAMID AIT ABDELLAHMAINE, Khalifa University, HOI DICK NG, GEORGIOS VATISTAS, Concordia University — The stirring flow driven by a rotating disk of a shallow water layer confined in a cylindrical bucket is revisited. The formation of a system of two and three satellite vortices, nested within slightly elliptical and triangular paraboloid free surface, orbiting around the center of the disk, is observed. At critical disk speeds transitions between these two systems of satellite vortices occur. These transitions were imaged and the velocity fields at the free surface of the shallow water were obtained via particle image velocimetry (PIV) measurement. The nucleation or theihilation of the satellite vortex during the two transitions is discussed in relation with the eigenmodes of the vortex-patterns.
1:45PM L13.00001 The Shedding of Jupiters Red Flakes Does Not Mean It Is Dying
PHILIP MARCUS, UC Berkeley, PEDRAM HASSANZADEH, Rice University, MICHAEL WONG, IMKE DE PATER, AIDI ZHANG, UC Berkeley, JOSEPH BARRANCO, UCSF, DAVID LEE, Rice University — During 2019 the Great Red Spot (GRS) of Jupiter repeatedly shed large (100,000 km$^2$) chunks of itself as red flakes. Rather than the GRS “dying” as report in the popular press, we have a more benign hypothesis tested with 3D numerical simulations. There are 2 distinct boundaries of the GRS (n.b., neither of which is coincident with the boundary of its cloud cover): (1) the boundary of its potential vorticity (PV) anomaly, and (2) its last “closed streamline”. An isolated vortex has nested closed streamlines, both interior to it and exterior it. The latter circumscribe the vortex. However, an anti-cyclone embedded in an anti-cyclonic zonal shear only has exterior closed streamlines near the PV boundary. Farther from its PV boundary, it has “open streamlines” that circumscribe the planet, not the vortex. The last close streamline contains at least one stagnation point. We show that when there is large area between the last closed streamline and the PV boundary, vortices “fed” to the GRS merge with it. However, when that area is small, vortices fed to the GRS will be expelled at or near a stagnation point. Thus, our explanation of the of recent Red Flakes is that area between the PV boundary of the GRS and its last closed streamline has shrunk.

1:58PM L13.00002 Nonlinear Magnetosonic Periodic and Solitary Waves in a Magnetized Dusty Plasma
NIMARDEEP KAUR, NARESHPAL SINGH SAINI, Guru Nanak Dev University, Amritsar — An investigation of magnetosonic nonlinear periodic (cnoidal) waves is presented in a magnetized electron-ion-dust plasma having cold dust fluid with inertialess warm ions and electrons. The reductive perturbation method is employed to derive the Korteweg-de Vries equation. The magnetosonic cnoidal wave solution is derived using the Sagdeev pseudopotential approach under the specific boundary conditions. There is the formation of only positive potential magnetosonic cnoidal waves and solitary structures in the high plasma-b limit. The findings of the present investigation may be helpful in describing the characteristics of various nonlinear excitations in Earth’s magnetosphere, solar wind, Saturn’s magnetosphere, and space/astrophysical environments, where many space observations by various satellites confirm the existence of dust grains, highly energetic electrons, and high plasma-$\beta$.

2:11PM L13.00003 Quantitative Flow Field Measurements of Astrophysical Relevance on the Blast-Driven Instability (RMI & RTI)$^1$ . SAMUEL PETTER, BENJAMIN MUSCI, GOKUL PATHIKONDA, DEVESH RANJAN, Georgia Institute of Technology — The presented work focuses on the implementation of Particle Image Velocimetry (PIV) to study the Blast-Driven Instability (BDI) in cylindrical geometry at the Georgia Tech Shock Tube and Advanced Mixing Laboratory. The facility uses detonators to generate a blast wave that accelerates the flow through a diverging test-chamber. The blast wave then interacts with a gaseous, membrane-less, interface of differing density, causing the occurrence of the combined Richtmyer-Meshkov (RMI) and Rayleigh-Taylor Instabilities (RTI); the two instabilities comprising the BDI. Previous validation of the facility was completed using high speed Mie Scattering and demonstrated faithful reproduction of the BDI phenomena. This validation garnered information about the qualitative development of the instability and identified aspects of improvement within the facility, both of which will be covered in this presentation. Preliminary PIV results are shown to corroborate the earlier Mie Scattering findings as well as predictions made by Taylor-Sedov theory derived from experimental pressure data.

$^1$DOE Early-Career Award, and DOE NNSA SSGF

2:24PM L13.00004 Horizontal shear instabilities in stellar radiative zones$^1$ . JUNHO PARK, VINCENT PRAT, STPHANE MATHIS, CEA Paris-Saclay (DRF/IRFU/DAP) — Shear flows in stratified-rotating fluids have been a popular research topic in astrophysics due to their applications for stellar evolution modeling. In stellar radiative zones, the thermal diffusivity is high with small Prandtl number of order $10^{-5}$. Also, the horizontal rotation component in latitudinal direction has not been generally considered with the traditional f-plane approximation, but recent research has revealed that it modifies significantly dynamics of inertia-gravity waves in the radiative zones. In this presentation, we revisit the horizontal shear instability problem by considering two components: the thermal diffusivity and the complete Coriolis acceleration rotation (i.e. the non-traditional f-plane approximation). And we study their impacts on hydrodynamic instabilities: inflection-point and inertial instabilities. With numerical and asymptotic stability analyses, we will present new results with mathematical formulations how the thermal diffusivity and horizontal modify these instabilities. For instance, a fast thermal diffusion destabilizes the inertial instability due to the suppression of the stratification effect while the horizontal rotation promotes instabilities and broadens the unstable regime.

$^1$The authors acknowledge support from the European Research Council through ERC grant SPIRE 647383.

2:37PM L13.00005 Dynamics in the Ball: Models for Fully Convective, Rotating M-Dwarf Stars . BENJAMIN BROWN, University of Colorado, JEFFREY OISHI, Bates College, DANIEL LECOANET, Princeton University, KEATON BURNS, Massachusetts Institute of Technology, GEOFFREY VASIL, University of Sydney — M-dwarf stars are smaller and less luminous than our Sun. In their interiors, convection dominates energy transport from the center of the star to their surface. This ball-like geometry is unique among all the stars on the main-sequence; in our Sun, solar convection is bounded from below by regions of stable stratification, creating a shell-like geometry instead. Within stellar convection zones, the turbulent plasma motions act as a dynamo, stretching and amplifying magnetic fields. M-dwarf stars have abundant and strong magnetic fields at their surfaces, but at a fundamental level we do not know whether these ball-like stars are similar to or different from our shell-like Sun. Here, using the novel spherical Dedalus pseudospectral framework, we consider the properties of convection and magnetic dynamo action in rotating, stratified simulations within global ball domains that capture the coordinate singularity at the center ($r = 0$), as well as the north and south pole. We find that global shearing flows are built by the convection, and these amplify global magnetic fields. Many ingredients in the fully convective M-dwarf simulations are similar to those found in simulations of the solar dynamo; this implies that these dissimilar stars may have similar internal processes.
2:50PM L13.00006 Radiative damping of convectively-driven gravity waves in the atmospheres of hot Jupiters1, JHETT BORDWELL, BENJAMIN BROWN, University of Colorado Boulder, JEFFREY OISHI, Bates College, WHITNEY POWERS, University of Colorado Boulder — Jovian atmospheres consist of a substantial, deep convection zone underlying a stably stratified region populated by convectively driven waves at many scales. These waves are significant to the pumping of large scale atmospheric jets, upper atmosphere heating, and chemical transport. To understand the role that radiation plays in the propagation of these waves, we perform numerical experiments with Dedalus at small scales studying wave driving and chemical transport in an atmosphere with radiative diffusion (appropriate for a hot Jupiter). We find that assuming an opacity structure appropriate for a Jovian atmosphere, all but the smallest of these waves are damped by radiative diffusion. We further compare our results with those of a Reynolds’ stress forcing model of wave driving, and explore the transport of reactive passive tracers through a simple Newtonian relaxation model.

1The authors acknowledge the support of the NASA Solar System Workings program (Grant 80NSSC19K0026)

3:03PM L13.00007 Nonlinear dynamics of forced baroclinic critical layers1, CHEN WANG, NEIL BALMFORTH, University of British Columbia — Baroclinic critical levels are singularities of waves propagating in inviscid stratified shear flow, and they play a crucial role in the self-replication of ’zombie vortices’. Our previous work has shown that for baroclinic critical layers under continuous forcing, the linear evolution features secular growth of density and decreasing thickness, and the following nonlinear evolution is characterized by a jet-like mean flow, which is focused exponentially at later times. In the present work, we show that thermal diffusion can arrest the focussing. A coherent structure of density is formed instead, which drifts in the cross-stream direction and leaves behind a growing defect in the mean-flow velocity. We explore detailed properties of the drifting coherent structure, and conjecture that it could be responsible for the expansion of critical layers as observed in the zombie vortices.

1C.W. thanks the University of British Columbia for a four-year doctoral fellowship.

Monday, November 25, 2019 1:45PM - 3:16PM
Session L14 Energy: Hydropower and General

1:45PM L14.00001 Influence of sea bed slope on the performance of a shore-fixed oscillating water column wave energy converter1, PIYUSH MOHAPATRA, TRILOECHAN SAHOO, ANIRBAN BHATTACHARYYA, Indian Institute of Technology Kharagpur — The oscillating water column (OWC) devices have been quite effective in harnessing energy from the ocean waves at prototype scales. A numerical study was carried out to analyze the influence of sea bed slope on the hydrodynamic efficiency of such a device. A computational fluid dynamics (CFD) based numerical wave tank (NWT) was developed using ANSYS-Fluent which uses a multiphase volume of fluid (VOF) method to simulate the ocean waves. The power take-off (PTO) unit of the device is modeled as a porous jump in the flow field to impose the pressure jump versus flow characteristics of the turbine. The sea bed beneath the OWC chamber is varied in accordance with slope height and slope length. The fluid flow parameters inside the chamber and the hydrodynamic efficiency of the chamber are calculated for various sloping bed conditions. The efficiency is plotted against the incident wave frequency and it was observed that the efficiency curve reaches a maximum for a certain incoming wave frequency. It was observed that depending on the sea bed slope not only the value of the peak efficiency varies but also there is a phase shift in the efficiency plot. It was verified that for a few of the slopes there is an improvement in hydrodynamic capture efficiency.

1Indian Institute of Technology Kharagpur

1:58PM L14.00002 Characterization of a Mangrove-Like System for Tidal Energy Harvesting: Effect of root and path diameter.1, DANIEL O. GOMEZ VAZQUEZ, Undergraduate Student, Universidad Ana G. Mendez, EDUARDO E. CASTILLO CHARRIS, Assistant Professor, Universidad Ana G. Mendez, AMIRKHOSRO KAZEMI, Post Doctoral Research Associate, Florida Atlantic University, OSCAR CURET, Assistant Professor, Florida Atlantic University — Tidal energy is a potential source of renewable energy for many coastal locations in the US. However, many of the conventional devices to convert hydrokinetic energy to electrical energy are not necessarily apt in these locations. Inspired by mangrove roots - a tree that is abundant along the tidal stream in tropical and sub-tropical areas, we designed a novel device to harvest hydrokinetic energy from tidal currents. This device consists of one or multiple oscillating cylinders, partially submerged in the water and an electric generator composed of fixed magnets and a winding. A steel plate provides a restoring spring force for the system. The device was tested in a recirculating water tunnel for different cylinder diameters, array sizes and flow speeds. The kinematics and the voltage output were measured for the different conditions. The flow downstream the model was measured using Particle Image Velocimetry. In general, the device starts to oscillate at a critical flow speed and reach a maximum velocity until the motion decreases. The maximum amplitude oscillation and voltage output tend to increase with larger cylindrical diameters and when the cylinders were closer to each other. The power performance and wake structure are also compared.

1 National Science Foundations Research Experience for Undergraduates program (EEC-1659468)

2:11PM L14.00003 Energy Harvesting from Mangrove-like Structure in Tandem and Staggered Arrangements.1, EDUARDO E. CASTILLO, Assistant Professor, Universidad Ana G. Mendez, DANIEL O. GOMEZ VAZQUEZ, Undergraduate Student, Universidad Ana G.Mendez, AMIRKHOSRO KAZEMI, Post Doctoral Research Associate, Florida Atlantic University, OSCAR CURET, Assistant Professor, Florida Atlantic University — This work investigates the performance of a mangrove-inspired structure that uses vortex induced-vibration to harvest hydrokinetic energy. The device consists of a coil and magnets attached to a vertical wood cylinder submerged in water and connected to a thin steel plate. This cantilever configuration allows only oscillations perpendicular to the flow. Three independent devices were placed in tandem and staggered arrangement in a water tunnel to measure the power generation and the kinematics of the three devices. Reynolds numbers ranged from 200 to 1500, based on cylinder diameter. It was found that the energy generated was proportional to the oscillation’s amplitude. The results show that in both arrangements as the velocity of the flow increases, the amplitude of oscillation increases from zero to a region with high values to then decrease to zero for high velocities. The up-stream device shows a delay in that behavior, compare to the other devices, but reach higher frequency of the oscillations. Additionally, we measured the flow structure to explore the hydrodynamic interaction within the devices. These renewable energy devices could have applications to power small actuators or sensors to monitor coastal infrastructure.

1 National Science Foundations Research Experience for Undergraduates program (EEC-1659468)
2:24PM L14.00004 Numerical simulation of a horizontal axis tidal turbine with a passive load-control system, WEIDONG DAI, IGNAZIO MARIA VIOLA, University of Edinburgh, RICCARDO BROGLIA, CNR.Institute of Italian Ship Model Basin Insein — Load fluctuations on Horizontal Axis Tidal Turbines (HATT) may result in fatigue failures, and this is one of the main factors that affect the reliability and durability of tidal turbines. While passive load-control systems for horizontal axis wind turbine have been well studied, there are few studies about such systems for tidal turbines. We propose a passive pitching mechanism to mitigate the change of angle of attack and, in this way, to lower the fatigue loading. We studied the fluid mechanics of a HATT with and without the passive pitching mechanism in open channel flow conditions. The flow field around the blades and forces on the blades were computed numerically both with a commercial and an in-house finite-volume CFD code, and validated with experimental data. We found that the use of passive pitching mechanism can reduce the amplitudes of thrust fluctuations by around 80% without affecting the mean power generated. We also observed that this mechanism can mitigate the fluctuation of power output by around 20%.

1This project is supported by Lanyang Geothermal Corp., Taiwan

2:37PM L14.00005 A total-flow design of geothermal power generator using Turgo turbines1, TZU-YUAN LIN, HSIEH-CHEN TSAI, National Taiwan University — We combine Turgo turbines and two-phase supersonic nozzles to design a total-flow geothermal power generator. The high-pressure subcooled liquid from the well forms supersonic flashing jets through the nozzles and the high-speed two-phase jets impinge turbine blades obliquely to drive the Turgo turbine. This new generator converts geothermal power directly from the two-phase fluid without implementing any phase separator or heat exchanger, which results in a simple and easy-to-maintain system. Compared with traditional Organic Rankine Cycle (ORC) generators, results from theoretical analysis and field tests in Yilan, Taiwan show that the new design of geothermal power generator has a competitive geothermal efficiency operating at moderate reservoir enthalpy.

1This work was supported by DOE EPSCoR GrantDE-SC0012671 under administration of Dr. Timothy Fitzsimmons.

2:50PM L14.00006 ABSTRACT WITHDRAWN —

3:03PM L14.00007 CFD Simulations of Floating Offshore Wind Turbine Platform1, MOSTAFA ELASKALANY, MAYSAM MOUSAVIRAAD, University of Wyoming — Computational modeling of offshore wind turbines imposes significant challenges due to the multiphysics nature of the system that involves both the turbine and the support structure. To provide reliable experimental data for validation of simulation tools, the OC5 DeepWind floating semi-submersible wind system was designed and tested through DOE support. The purpose of this work is to develop and validate a high-fidelity computational solver for floating offshore platforms that is capable of modeling incident waves, sea current, hydrodynamics, mooring dynamics, and foundation dynamics. The high-performance fluid solver is based on URANS CFD methods, capable of modeling incoming waves, including stochastic/random ocean waves based on specified spectrum, and resolves the nonlinear wave dynamics. Validation studies are presented for free decay as well as wave-only excitation in regular waves free to surge, heave, and pitch motions. Further computational studies are carried out in unidirectional irregular stochastic waves, including uncertainty quantification (UQ) for random input wave parameters. Future studies will include coupling with fully resolved turbine blades modeling, as well as stochastic optimization studies for improved hydrodynamics and aerodynamics.

Monday, November 25, 2019 1:45PM - 3:16PM — Session L15 Turbulent Boundary Layers: Curvature and Pressure Gradients II

1:45PM L15.00001 Non-Equilibrium Boundary Layer Development in Response to Changing Pressure Gradients1, RALPH VOLINO, U.S. Naval Academy — Experiments were conducted in non-equilibrium turbulent boundary layers on a smooth flat wall. The test section included a zero pressure gradient (ZPG) development region, followed by a favorable pressure gradient (FPG) region with constant acceleration parameter $K = (\nu/U_{\infty}^2)(dU_{\infty}/dx)$, a ZPG recovery, and a constant $K$ adverse pressure gradient (APG) region. Eight cases were included with three different inlet velocities and $K$ in the FPG ranging from $0.125 \times 10^{-6}$ to $2 \times 10^{-6}$. The $K$ magnitude of the APG in each case was half that of the FPG. Velocity profiles were acquired at 12 streamwise stations using a two component LDV, and PIV was used to document velocity fields at the same stations. The mean velocity and turbulence profiles and correlations were observed to change in response to the changing pressure gradients, and changes in various quantities (e.g. wake strength or $u''$ at representative $y^+$) were quantified as functions of streamwise location. Normalizations of streamwise distance were found for the FPG, ZPG, and APG regions that caused the data to collapse, quantifying the non-equilibrium response to the changing pressure gradients.

1Sponsored by the Office of Naval Research

1:58PM L15.00002 Spatio-temporal dynamics of pressure-induced turbulent separation bubbles1, WEN WU, CHARLES MENEVEAU, RAJAT MITTAL, Johns Hopkins University — The spatio-temporal dynamics of separation bubbles (SBs) induced to form in a fully-developed turbulent boundary layer over a flat plate are studied via direct numerical simulations. Two different SBs are examined: one induced by a suction-blowing velocity profile on the top boundary and the other, by a suction-only velocity profile. The latter condition allows reattachment to occur without an externally imposed favourable pressure gradient and leads to a SB more representative of those occurring over airfoils and in diffusers. The suction-only SB exhibits a range of clearly distinguishable modes. The high-frequency modes are well characterized by classical frequency scalings for a plane mixing layer and are associated with the formation and shedding of spanwise oriented vortex rollers. The topology associated with the low-frequency motion is revealed by the application of dynamic mode decomposition (DMD) and is shown to be dominated by highly elongated structures in the streamwise direction. The possibility of Görtler instability as the underlying mechanism for these structures is explored.

1The authors acknowledge support from AFOSR Grant FA9550-17-1-0084
2:11PM L15.00003 Effects of Adverse Pressure Gradient and Convex Curvature on the Evolution of Turbulence Statistics and Coherent Structures. SIMERET GENET, LIUYANG DING, ALEXANDER SMITS, MARCUS HULTMARK, Princeton University — Here we are interested in developing models and scaling relationships that better characterize non-equilibrium, turbulent, wall-bounded flows. In this study, an axisymmetric body is used to introduce streamwise pressure gradients and streamline curvature to a fully developed turbulent flow in a pipe. This body consists of three sections which are the bow, recovery region and the stern. The flow first sees the bow that creates a favorable pressure gradient. It then passes over the recovery region before meeting the stern that introduces an adverse pressure gradient (APG). Particle Image Velocimetry (PIV) is used to measure the flow field. The evolution of turbulent statistics as well as deformations of coherent structures are quantified in the stern region of the body of revolution. Different sized bodies are used to vary the strengths of the APG. Lastly, results from PIV are compared with data obtained from hot wire measurements at high Reynolds numbers in the Superpipe.

1 ONR Grant N00014-17-1-2309

2:24PM L15.00004 Mean Momentum Balance in Adverse Pressure Gradient Boundary Layers. SYLVIA ROMERO, SPENCER ZIMMERMAN, JIMMY PHILLIP, JOSEPH KLEWICKI, University of Melbourne — Utilizing Large Eddy Simulation (LES) data from Bobke et al. (2017) the mean momentum equation for a two-dimensional adverse pressure gradient boundary layer (APG BL) will be analyzed in comparison to the channel flow case. The properties of the mean momentum balance (MMB) for a channel flow are well characterized (e.g. see Wei et al. (2005)). Like the channel flow case, the MMB for the two-dimensional zero pressure gradient boundary layer (ZPG BL) is a balance of three terms. Morrill-Winter et al. (2017) transformed the MMB terms of the ZPG BL into a form like the channel flow case. Using the data of Bobke et al. (2017), the present analysis examines whether a similar transformation is applicable to APG flows. The MMB for APG flows is a balance of four terms, unlike the three for the ZPG BL and channel flow. The four terms have physical interpretations that are tied to the flow physics. The change in dominance of the MMB terms will be discussed, as well as the dependence of the balance on the pressure gradient and distance from the wall. Various aspects such as the history effects of the developing pressure gradient and the location where the viscous force loss dominance will also be discussed.

1 Office of Naval Research: N00014-17-1-2307 and Australian Research Council

2:37PM L15.00005 Direct Numerical Simulations of Separated Flow over a Bump. RICCARDO BALIN, University of Colorado Boulder, PHILIPPE SPALART, The Boeing Company, KENNETH JANSEN, University of Colorado Boulder — A turbulent boundary layer over a Gaussian bump is computed by direct numerical simulation (DNS) of the incompressible Navier-Stokes equations. The two-dimensional bump causes a rapid succession of favorable-to-adverse pressure gradients that can lead to shallow separation on the downstream side, both of which are characteristic of the flow over the flap of an aircraft wing in high-lift configurations. At the inflow, the momentum thickness Reynolds number is approximately 1,000, and the boundary layer thickness is 1/8 of the bump height. Results from DNS are discussed, describing the complex flow physics observed and the effects of variations to the inflow mean profile. Data from this study will be used for the development of lower fidelity turbulence models through data driven approaches.

1 This work is supported by the National Science Foundation (NSF), award number CBET-1710670, and by the National Aeronautics and Space Administration (NASA), award number 80NSSC18M0147. Resources of the Argonne Leadership Computing Facility, a DOE Office of Science User Facility, and of the NASA HECC facilities were used.

2:50PM L15.00006 Validation Experiments for Turbulent Separated Flows over Axysymmetric Afterbodies. SAMANTHA GILDERSLEEVE, CHRISTOPHER L. RUMSEY, NASA Langley Research Center — Historically, the flow physics involved with most turbulent separated flows have presented fundamental challenges in validation between experimental and numerical approaches. As recognized by the CFD Vision 2030 study commissioned by NASA, validation of RANS models and scale-resolving methods for turbulent flows requires the support of advanced, high-fidelity experiments designated specifically for CFD implementation. In accordance with this effort, a new test platform, referred to as the NASA Axysymmetric Afterbody, was designed to obtain detailed information in a flow with a smooth, adverse-pressure gradient induced separation. The parametric body offers a range of flow behaviors from fully attached, incipient separation, to fully separated flow at subsonic conditions. In an initial effort to validate CFD capabilities, surface pressure measurements and flow visualizations at Re=180,000 are compared against several RANS turbulence models to understand the separation behaviors and identify critical variability between the models. Results indicate potential discrepancies due to the effect of Reynolds number and physical tunnel blockage. Ongoing work will continue the experimental campaign to obtain off-body flow measurements using SPIV and LDV techniques to yield further insight.

3:03PM L15.00007 The law of incipient separation for turbulent flows as inferred by RANS. DAWEI LU, ABHIRAM AITHAL, ANTONINO FERRANTE, University of Washington, Seattle — We have performed Reynolds-averaged Navier-Stokes (RANS) computations of incompressible turbulent boundary layer with adverse pressure gradient over two-dimensional smooth curved ramps of length L for 982 ≤ Re₉ ≤ 3698. First, we have validated the RANS by comparing the results with the experiments of Song & Eaton (Expts. Fluids, 2004) for the separated flow over an arc at Re₉ = 1100. Then, we have investigated the effects of the ramp slope and curvature on the skin-friction and pressure coefficients. Our results show a numerical criterion of incipient flow separation determined only by the geometrical parameters of maximum slope, |z'|max, length, L, and height, h, of the ramp, and by Re₉. Specifically, incipient separation occurs when |z'|max normalized by h/L (|z'|max = |z'|max/hL) reaches a critical value that is determined according to the following law: |z'|crit = αRe₉⁻¹/θ + β, where α < 0 and β > 0 are constants. Accordingly, the flow separates over a curved smooth ramp for which |z'|max > |z'|crit. The uncertainty of the law is also reported.

Monday, November 25, 2019 1:45PM - 3:16PM
Session L16 Focus Session: Exascale Computations of Complex Turbulent Flows II
4c3 - Fady Najjar
Applications, and report progress testing its implementation for relevant canonical problems. Test cases include the Taylor-Green vortex closures. We propose a dynamic blended hybrid RANS/ILES bridging strategy for applications involving variable-density turbulent mixing promoted by energy deposited at the material interfacial layers during the shock interface interactions. Transition involves unsteady large-scale multiscale resolution issues of shocks and variable density turbulence, we must address the difficult problem of predicting flow transitions implicit LES (ILES), where the small-scale flow dynamics is presumed enslaved to the dynamics of the largest scales. Beyond the complex Confinement Fusion (ICF) capsule implosions. Such complex flow physics is capturable with coarse grained simulation (CGS) – classical and from perturbations at shocked material interfaces, as vorticity is introduced by the impulsive loading of shock waves – e.g., as in Inertial GERMANO, Duke U. — We focus on simulating the consequences of material interpenetration, hydrodynamical instabilities, and mixing arising of the thermal scales within the flow.

were able to identify the destruction of turbulent convection caused by radiative damping of thermal fluctuations and relate it to the dimension level Regent programming language, which is itself a high level counterpart to the Legion programming system. We will give an overview of the whole description of TRI in a direct numerical simulation framework. We applied the algorithm to a thermally developing turbulent channel and the radiative transfer equation is solved on GPUs using an optimized Monte Carlo method. With our method it is possible to access the solution of the radiative transfer problem on fine grids is notoriously challenging, especially for optically intermediate systems; we implemented so called turbulence-radiation interactions (TRI) greatly modify the well-known patterns of heat transfer and variable property turbulence. The Delft University of Technology — When dealing with high temperature applications, thermal radiation plays an important role in the heat transfer process. In particular, due to its non-locality, radiation causes counter-intuitive interactions with the turbulent temperature field. These so called turbulence-radiation interactions (TRI) greatly modify the well-known patterns of heat transfer and variable property turbulence. The solution of the radiative transfer problem on fine grids is notoriously challenging, especially for optically intermediate systems; we implemented an innovative approach which exploits heterogeneous high performance computing facilities. The Navier-Stokes equations are solved on CPUs, and the radiative transfer equation is solved on GPUs using an optimized Monte Carlo method. With our method it is possible to access the whole description of TRI in a direct numerical simulation framework. We applied the algorithm to a thermally developing turbulent channel flow of high temperature water vapor to study the interaction between the different heat transfer mechanisms. With the obtained results we were able to identify the destruction of turbulent convection caused by radiative damping of thermal fluctuations and relate it to the dimension of the thermal scales within the flow.

Direct Numerical Simulation of Coupled Convection and Radiation on Heterogeneous Computing Architectures. SIMONE SILVESTRI, RENE PECNIK, DIRK ROEKAERTS, Delft University of Technology — When dealing with high temperature applications, thermal radiation plays an important role in the heat transfer process. In particular, due to its non-locality, radiation causes counter-intuitive interactions with the turbulent temperature field. These so called turbulence-radiation interactions (TRI) greatly modify the well-known patterns of heat transfer and variable property turbulence. The solution of the radiative transfer problem on fine grids is notoriously challenging, especially for optically intermediate systems; we implemented an innovative approach which exploits heterogeneous high performance computing facilities. The Navier-Stokes equations are solved on CPUs, and the radiative transfer equation is solved on GPUs using an optimized Monte Carlo method. With our method it is possible to access the whole description of TRI in a direct numerical simulation framework. We applied the algorithm to a thermally developing turbulent channel flow of high temperature water vapor to study the interaction between the different heat transfer mechanisms. With the obtained results we were able to identify the destruction of turbulent convection caused by radiative damping of thermal fluctuations and relate it to the dimension of the thermal scales within the flow.

Dynamic Bridging Modeling for Coarse Grained Simulations of Shock Driven Turbulent Mixing. FERNANDO GRINSTEIN, JUAN SAENZ, RICK RAUENZAHN, LANL, MASSIMO GERMANO, Duke U. — We focus on simulating the consequences of material interpenetration, hydrodynamical instabilities, and mixing arising from perturbations at shocked material interfaces, as vorticity is introduced by the impulsive loading of shock waves — e.g., as in Inertial Confinement Fusion (ICF) capsule implosions. Such complex flow physics is capturable with coarse grained simulation (CGS) – classical and implicit LES (ILES), where the small-scale flow dynamics is presumed enslaved to the dynamics of the largest scales. Beyond the complex multiscale resolution issues of shocks and variable density turbulence, we must address the difficult problem of predicting flow transitions promoted by energy deposited at the material interfacial layers during the shock interface interactions. Transition involves unsteady large-scale coherent-structure dynamics resolvable by CGS but not by RANS modeling based on equilibrium turbulence assumptions and single-point-closures. We propose a dynamic blended hybrid RANS/ILES bridging strategy for applications involving variable-density turbulent mixing applications, and report progress testing its implementation for relevant canonical problems. Test cases include the Taylor-Green vortex – prototyping transition to turbulence, and a shock tube experiment – prototyping shock-driven turbulent mixing.

Large-eddy simulation of Rayleigh-Taylor mixing on the Sierra supercomputer. BRANDON MORGAN, JASON BURMARK, MICHAEL COLLETTE, CYRUS HARRISON, MATTHEW LARSEN, BRIAN PUBLINER, BRIAN RYUJIN, Lawrence Livermore National Laboratory — The Sierra system is Lawrence Livermore National Laboratory’s first production supercomputer accelerated by graphics processing units (GPUs). As part of the system’s initial acceptance testing in October 2018, large-eddy simulation was conducted of Rayleigh-Taylor mixing in a spherical geometry using 97.8 billion computational volumes across 16,384 GPUs on Sierra. This talk will discuss how the Sierra system enabled such a massive calculation and how the results have been used to inform development of the k-L-a-Y1 Reynolds-averaged Navier-Stokes (RANS) model for reacting turbulence [Morgan, B. E., Olson, B. J., Black, W. J., and McFarland, J. A., “Large-eddy simulation and Reynolds-averaged Navier-Stokes modeling of a reacting Rayleigh-Taylor mixing layer in a spherical geometry.” Phys. Rev. E 98, 033111 (2018)].

This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.
2:50PM L16.00006 Towards Exascale Direct Numerical Simulations of Multi-Stage Ignition and Turbulent Mixing in Diesel Jets. JACQUELINE CHEN, MARTIN RIETH, MYOUNGKE LEE, Sandia National Laboratories, ELLIOTT SLAUGHTER, SESHU YAMAJALA, SLAC Accelerator National Laboratory, ALEX AIKEN, Stanford University — Direct numerical simulations of multi-stage ignition in low-temperature surrogate diesel jets is used to study ‘turbulence-chemistry’ interactions governing cool flame propagation and turbulent diffusion and their role in accelerating low- and high-temperature ignition. The effects of varying the ambient temperature and oxygen concentration on mixture formation and combustion processes is quantified. Conditional statistics are presented showing the significance of turbulent diffusion relative to laminar flame propagation. These simulations are enabled by an asynchronous task-based programming model and runtime, Legion, which is used to obtain scalable performance of the Legion-S3D DNS code on Summit at the Oakridge Leadership Computing Facility. The Legion runtime is able to hide the memory latency by overlapping communication and computation and to optimize data movement. Refactoring Legion-S3D in Regent, a companion compiled language that maps directly onto the Legion runtime, simplifies and enforces the rules of the Legion programming model enabling domain scientists to write extensible code with sequential semantics.

1sponsored by DOE ASCR and BES

3:03PM L16.00007 Parallel and dynamic mesh adaptation of tetrahedral-based meshes for propagating fronts and interfaces: application to premixed combustion and primary atomization. VINCENT MOUREAU, PIERRE BENARD, GHISLAIN LARTIGUE, CORIA, CNRS UMR6614, Normandie University, INSA et Université de Rouen, France — Thanks to the steady growth of computational resources and a large effort on solver optimization, Large-Eddy Simulation (LES) of realistic systems has become attainable. In these systems, turbulent multi-physics flows involve a large range of scales that need to be resolved by the mesh to capture the proper flow dynamics. Adaptive or dynamic mesh adaptation (AMR) is an appealing technique to reduce the modeling errors in LES. AMR of tetrahedral-based meshes for LES is difficult as it requires numerous mesh topology changes and high-quality grids to resolve the turbulent scales that are close to the cut-off frequency of the mesh. A parallel AMR strategy has been developed recently [Bnard et al., IJNMF 2015] in the YALES2 flow solver [www.coria-cfd.fr]. It combines adaptation and repartitioning steps to enable the AMR of massive grids counting billion cells exploiting up to tens of thousand cores. Mesh adaptation relies on the work of Dapogny et al. [JCP 2014] available in the MMG library [www.mmgtools.org]. The presentation will focus on the parallel adaptation strategy for both volume and surface meshes, its optimization on modern super-computers and on various academic and industrial applications related to premixed turbulent combustion and primary atomization.

Monday, November 25, 2019 1:45PM - 3:16PM –
Session L17 Focus Session: Recent Advances in Data-driven and Machine Learning Methods for Turbulent Flows III

1:45PM L17.00001 Machine learning meets mechanism: Mechanism of roll reversal in Rayleigh-Bénard Convection, XI CHEN, XIAOJUE ZHU, MICHAEL BRENNER, Harvard University — Understanding the precise physical events that underlie the reversal of circulation direction in Rayleigh-Bénard (RB) convection has long been a mystery. We aim to solve this problem by using a machine learning model to classify the events related to a reversal in convection. We found a unique time scale indicating the transition between two circulation directions of RB-Flow, which is much shorter than the periodicity of reversals. We then try to invert the neural network, to discover the precise flow events that cause the physics on this timescale. This allows us to identify local patterns of the flow field that are critical to the reversal phenomena. We use these as the basis for building a phenomenological theory of reversal. We believe using interpretable machine learning in this way can be applied for the study of other fluid dynamics problems and even a lot of science problems.

1:58PM L17.00002 Physics-Informed Echo State Networks for the Prediction of Extreme Events in Turbulent Shear Flows. NGUYEN ANH KHOA DOAN, WOLFGANG POLIFKE, Technical University of Munich, LUCA MAGRI, University of Cambridge — A large number of turbulent flows exhibit extreme events. Extreme events are here defined as large-amplitude deterministic events, which suddenly occur aperiodically in the chaotic attractor. The time-accurate prediction of extreme events is challenging because of (i) the butterfly effect, which is the main property of the chaotic dynamics of turbulent flows, and (ii) the unpredictable nature of extreme events. We develop a physics-informed data-driven framework, the Physics-Informed Echo State Network (PI-ESN), to predict extreme events in turbulent flows. The PI-ESN consists of a reservoir of dynamical neurons, which learn the systems dynamics from time-series of its evolution. We apply this method to a turbulent shear flow between two free-slip walls subject to a sinusoidal force. This flow displays abrupt transition between quasi-laminar and fully-turbulent states. We are able to time-accurately predict the flow evolution during these extreme events by imposing the physical principles as constraints in the learning algorithm. This physics-informed data-driven approach outperforms purely data-driven approaches, which opens up new horizons for the time-accurate prediction of turbulent flows by leveraging on data and physical principles.

The support of TUM-IAS, funded by the German Excellence Initiative and the EU 7th Framework Programme under grant agreement no. 291763 is acknowledged. LM also acknowledges the Royal Academy of Engineering Research Fellowship Scheme.

2:11PM L17.00003 Data-driven prediction of vortical structures in turbulent flows employing deep learning techniques, BABAK KASHIR, MARCO RAGONE, VITALIY YURKIV, FARZAD MASHAYEK, University of Illinois at Chicago — The vortical structures are inherent characteristics of turbulent flows. Significant research has been conducted to understand, characterize and locate the vortical structures in turbulent flows in complex configurations. However, the identification of the size and location of the vortices in practical flows is often challenging due to the interplay of various parameters. To overcome these challenges, we have developed a deep learning model to identify and locate the vortical structures. The deep learning model is trained and tested in lid-driven cavity flow fields to predict the vortical structures in different conditions. The architecture of the model is characterized by multiple layers with random initialization and linear regularization, whereas the final prediction is performed through a binary classification. The second invariant of the velocity gradient tensor (known as the Q-criterion) is used to locate the vortical structures in fluid dynamics. This criterion describes a vortex as a continuous fluid region with the positive second invariant. The neural-network predictions are compared with the results from the previously validated numerical simulations. The present study allows for advance accelerated analysis of complex turbulent flows.

1The support of TUM-IAS, funded by the German Excellence Initiative and the EU 7th Framework Programme under grant agreement no. 291763 is acknowledged. LM also acknowledges the Royal Academy of Engineering Research Fellowship Scheme.
A priori translational invariance and different inputs are tested aiming to provide a Galilean invariant model. A convolution operation ensures of the local three-dimensional (3D) strip which provides a two-dimensional formulation with additional statistical terms accounting for the 3D.

We propose a new flow decomposition based on the spanwise average fluctuations and sound level spectrum of pressure fluctuations are successfully obtained to assess flow and noise characteristics of side mirror models.

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A1B2007117).

2:37PM L17.00005 Deep learning the spanwise-averaged turbulent wake of a circular cylinder1, BERNAT FONT GARCIA, Univ. of Southampton and Inst of High Performance Computing (A*STAR), GABRIEL WEYMOUTH, University of Southampton, VINH-TAN NGUYEN, Institute of High Performance Computing (A*STAR), OWEN TUTTY, University of Southampton — Numerical simulations of long and flexible cylindrical structures become prohibitive at high Reynolds regimes because of the wide range of spatial and temporal scales that need to be resolved. We propose a new flow decomposition based on the spanwise average of the local three-dimensional (3D) strip which provides a two-dimensional formulation with additional statistical terms accounting for the 3D.

Numerical simulations of long and flexible cylindrical structures become prohibitive at high Reynolds regimes because of the wide range of spatial and temporal scales that need to be resolved. We propose a new flow decomposition based on the spanwise average of the local three-dimensional (3D) strip which provides a two-dimensional formulation with additional statistical terms accounting for the 3D.

The equations highlight a symmetry of the pressure Hessian. Indeed, the component of the non-local pressure Hessian, reducing it to a tensor of rank two. This simplifies the geometric interpretation of the pressure Hessian effect on the dynamics of the velocity gradient invariants and allows to compare two- and three-dimensional flows. We characterize the geometry of the effective pressure Hessian by means of DNS results, focusing on the statistics of its independent eigenvalue and the alignment between the effective pressure Hessian and Navier-Stokes equations.

This work was supported by Samsung Research Funding Center of Samsung Electronics under Project Number SRFC-TB1703-01 and National Research Foundation of Korea (NRF) under Project Number NRF-2017R1E1A1A03070514.

2:50PM L17.00006 Mechanisms of convolutional neural networks for learning three-dimensional unsteady wake flow1, SANGSEUNG LEE, DONGHYUN YOU, Pohang University of Science and Technology — Recently, convolutional neural networks (CNNs) have been applied to predict or model flow dynamics. However, mechanisms of CNNs for learning flow dynamics are still not well understood, while such understanding is highly necessary to reduce trial-and-errors in designing networks. In the present study, we investigate the mechanisms of a CNN for prediction of three-dimensional unsteady wake flow behind a circular cylinder. Feature maps in the CNN are visualized to compare flow structures that the CNN extracts from flow at different flow regimes. A Fourier analysis is conducted to reveal the mechanisms, which enable the CNN to predict flow dynamics at different flow regimes, of a convolution layer to integrate and transport wave number information from flow. The integration and transportation characteristics of information of flow variables and histories in the CNN are discussed.

1This work was supported by Samsung Research Funding Center of Samsung Electronics under Project Number SRFC-TB1703-01 and National Research Foundation of Korea (NRF) under Project Number NRF-2017R1E1A1A03070514.

3:03PM L17.00007 Learning effective viscosity for moderate Reynolds number Navier-Stokes equations, XIAOJUE ZHU, MICHAEL BRENNER, Harvard University — We propose that with an appropriately chosen effective viscosity $\nu(Re)$, a linearization of the Navier-Stokes equations perfectly captures the drag-determining features of flows around unsteady translating bodies, in the Reynolds number of order hundreds. Two ways are implemented to find the effective viscosity. First, $\nu(Re)$ is determined so that the time-averaged shapes of separatrix, a fluid surface that delimits the compact region of fluid that is entrained by the moving point force, match as closely as possible between the linearization equation and the Navier-Stokes equation. Second, $\nu(Re)$ is learned from a data-driven method, i.e. a deep neural network, by minimizing the mean squared error loss. We compare the results for the two methods, and we find that the data-driven method dramatically outperforms the former traditional method. We apply the linearization to the classical problem of predicting vortex shedding around a cylinder.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A1B2007117).
1:58PM L18.00002 Velocity gradient (VG) decomposition into normal-strain, pure shear, solid-body-rotation tensors: new insights into turbulence VG dynamics, RISHTA DAS, SHARATH GIRIMAJI, Texas A&M University — Velocity gradient tensor (VGT, \(A_{ij}\)) decomposition into symmetric (\(S_{ij}\)) and anti-symmetric (\(W_{ij}\)) tensors is unable to segregate the effect of shear present in both the strain-rate and rotation-rate tensors. In this study, an additive decomposition of VGT into normal strain-rate (\(N_{ij}\)), solid body rotation (\(R_{ij}\)) and pure shear (\(H_{ij}\)) tensors, is employed. In this decomposition, shear is clearly demarcated from pure rotation and pure normal strain effects. Then, we use direct numerical simulation data of incompressible forced isotropic turbulence in Taylor Reynolds number range \(Re_{\lambda} \in (200 - 600)\), to examine statistical and dynamical aspects of velocity gradient dynamics. We investigate (i) the probability distribution of the magnitude (Frobenius-norm) of these pure strain, pure shear and solid-body rotation tensors; (ii) the preferential alignment of the axis of solid-body rotation with the normal strain-rate eigenvectors; (iii) the conditional statistics of different VG processes as a function of these tensors; and (iv) intermittency phenomenon. The study develops unique and novel insights into turbulence processes which are not evident from previous VGT decompositions.

2:11PM L18.00003 Area Rule for velocity circulation\(^1\), KARTIK IYER, KATEPALLI SREENIVASAN, New York University, P.K. YEUNG, Georgia Institute of Technology — The statistical theory of velocity circulation at high Reynolds numbers has witnessed renewed interest following recent studies, both empirical (Iyer et al., arXiv:1902.07326, 2019) and theoretical (Migdal, arXiv:1903.08613, 2019). A central tenet in the scaling theory of circulation is the Area Rule which states that the probability distribution of the circulation around closed contours, whose characteristic dimensions reside in the inertial range, depends solely on the minimal spanning surface of the contour. We examine the Area Rule for both low and high order circulation moments, for different contour shapes and sizes, using the DNS of the three-dimensional Navier-Stokes equations within a cube with periodic boundary conditions, with the integral-scale Reynolds number spanning over two decades. Useful comparisons with other fields such as the Gaussian random fields will be discussed to highlight the simplicity afforded by circulation in the statistical description of small-scale turbulence.

\(^1\)This work is supported by NSF grant 1640771. The computations were performed using supercomputing resources provided through the XSEDE consortium at the Texas Advanced Computing Center at the University of Texas (Austin) and the Blue Waters Project at the National Center for Supercomputing Applications at the University of Illinois (Urbana-Champaign).

2:24PM L18.00004 Intermittency of Incompressible Passive Vector Convected by Homogeneous Turbulence\(^2\), TOSHIYUKI GOTOH, JINGYUAN YANG, Nagoya Institute of Technology, HIDEAKI MIURA, National Institute for Fusion Science, TAKEHISI WATANABE, Nagoya Institute of Technology — It is known that the fluctuation of passive scalar convected by turbulence is stronger than that of the turbulent velocity. In order to understand the physical mechanism yielding this difference we have studied the fluctuations of an incompressible passive vector \(u\) convected by the isotropic turbulence by comparing them with the passive scalar. It is found that the passive vector spectrum obeys the Obukhov-Corssin spectrum \(k^{-5/3}\) with constant \(C = 0.09\) and the low order statistics is close to the velocity spectrum at large scales and resembles the passive scalar at small scales. Strength of the intermittency of the passive scalar is intermediate between the velocity and the passive scalar. The domain size of intense \(\nabla \times u^2\) is found to be sheet like and similar to the scalar gradient. It is argued that the linearity of the equation is the key to generate the stronger intermittency of the passive fields.

\(^2\)Supported through computing resources at OLCF (DOE INCITE 2017) and TACC (XSEDE).

2:37PM L18.00005 Toward investigation of local vortex line topology in turbulence, BAJIRANG SHARMA, RISHTA DAS, SHARATH GIRIMAJI, Texas A&M University — Current vortex identification methods employ \(Q\) or \(\lambda_2\) criterion to investigate vortical structures in turbulent flows. We propose an alternate method to examine the local vortex structure. Specifically, we construct the vorticity vector field (\(\zeta\)) and compute the vorticity gradient tensor \(J_{ik} \equiv \partial \omega_i / \partial x_k\). The tensor \(J_{ik}\), similar to the velocity gradient tensor \(A_{ij} \equiv \partial u_i / \partial x_j\), is trace free. In a manner similar to streamline topology analysis using the second and third invariants of \(A_{ij}\), the second (\(Q_2\)) and third (\(R_3\)) invariants of \(J_{ik}\) are used to develop the topological description of local vortex line structure. Such a representation is not only useful for investigating the local vortex-line structure but is also useful in identifying key turbulence phenomena such as vortex line reconnection. Direct numerical simulation (DNS) data of incompressible forced isotropic turbulence and perturbation-evolution in plane-Poiseuille flow are analysed in this study. We investigate the following: (i) distribution in these two key canonical turbulent flows; and (ii) the formation of hair-pin vortices in transitioning plane Poiseuille flow using the aforementioned method.

2:50PM L18.00006 Origin and implications of odd-spin contributions in rapidly distorted turbulence, SUSAN KURIEN, Los Alamos National Laboratory, TIMOTHY CLARK, University of New Mexico, ROBERT RUBINSTEIN, Retired — We present mathematical calculations and supporting data from numerical simulations to demonstrate the emergence of reflection-symmetry breaking along the polar axis in flows, in the rapid distortion limit. The mathematical decomposition of second-rank tensors (eg: velocity correlations in wavenumber space) is done in the SO3 basis. We show the appearance of symmetry breaking for various reflection-symmetric initial conditions including isotropic and axisymmetric turbulence. The strain-rate tensor used to achieve rapid distortion also remains symmetric in all our test cases. These results help to elucidate the mechanism by which the so-called odd-spin (reflection-symmetry breaking) terms arise in the SO3; they also clarify the separate role of this type of symmetry breaking from other explicitly reflection-symmetric forcings such as helical or rotational strains.

3:03PM L18.00007 How the ramp-cliff structures in scalar turbulence vary with the Schmidt number and how to model that variation\(^3\), DHAWAL BUARI, New York University, M.P. CLAY, Georgia Institute of Technology, K.R. SREENIVASAN, New York University, P.K. YEUNG, Georgia Institute of Technology — In turbulent mixing of passive scalars, the ramp-cliff structures observed in one-dimensional cuts of the scalar field are responsible for a few important consequences, such as, departure from local isotropy, saturation of scaling exponents with respect to the moment-order. These results are established under the case of unity Schmidt number (\(Sc\)) given by the ratio of the kinematic viscosity of the fluid to the scalar diffusivity. In this talk, our first goal is to show how the above results vary with increasing \(Sc\). We utilize a massive DNS database with the Taylor-scale Reynolds number in the range \(140 - 650\), and \(Sc\) in the range \(1 - 512\), for the case of passive scalar with a uniform mean scalar gradient, mixed by forced isotropic turbulence. In particular, we investigate how the odd moments of the scalar derivatives (which are the symptoms of the ramp-cliff structures) vary with \(Sc\). A model based on the changing ramp-cliff structures is presented to describe the observed scaling of the scalar derivative statistics. We also address the scaling of scalar increments for varying \(Sc\) and particularly explore the saturation of exponents with respect to the order of their moments.

\(^3\)Supported through computing resources at OLCF (DOE INCITE 2017) and TACC (XSEDE).
1:45PM L19.00001 Numerical multiscale methods and effective boundary conditions
SEAN CARNEY, BJORN ENGQUIST, ROBERT MOSER, University of Texas at Austin — Numerical homogenization refers to the numerical extraction of the effective, “macroscopic”, or large scale behavior of a complex dynamical system at a reduced cost to resolving the full dynamics at all levels of detail. The no-slip boundary condition (BC) for viscous fluid flow over a solid surface can introduce asymptotically small scales that pose severe challenges for simulation; in this case it can be preferable to replace the no-slip condition with a homogenized BC. This talk discusses numerical techniques for generating slip BC, or wall laws, for laminar flows over rough boundaries, as well as turbulent boundary layer flows for constant favorable, zero, or adverse \( \nabla \cdot \mathbf{v} \). Guided by rigorous mathematics in the former case and recent empirical advances in the latter, numerical strategies are presented to overcome the high computational cost of resolving the full near wall dynamics. In both settings, the main idea consists of running high resolution simulations in a relatively small domain localized to the boundary. Numerical examples presented throughout validate the modeling approach.

1:58PM L19.00002 Extending Partially-Averaged Navier-Stokes equations to Variable-Density Turbulent Flow
FILOPE PEREIRA, FERNANDO GRINSTEIN, DANIEL ISRAEL, RICK RAUENZAHN, Los Alamos National Laboratory, SHARATH GIRIMAJI, Texas A&M University — The mixing of distinct fluids is of importance to various areas of engineering. This class of problems is featured by its variable-density (VD) that leads to buoyancy effects, hydro-dynamical instabilities, transition and turbulent flow. All these phenomena turn the modeling of VD flows difficult. Whereas DNS and LES models are excessively demanding for practical problems, RANS tends to poorly predict such flows. Bridging methods, on the other hand, have the potential to surpass many of the limitations of the former models. By resolving only the phenomena that are not amenable to be modeled, these models can achieve a good compromise between accuracy and cost. Yet, their development for VD flows is rife with challenges. The aim of this work is to develop a Partially-Averaged Navier-Stokes (PANS) framework for VD flow. To this end, the PANS framework proposed by Girimaji (2005) is extended to VD flow in order to derive a PANS-BHR2 closure. Particular attention is paid to the selection of the physical resolution (range of resolved scales) of the model. Thus, apriori testing is conducted to propose guidelines toward the efficient selection of the parameters determining the physical resolution of PANS-BHR2. The proposed model is then evaluated on two archetypal flow problem.

2:11PM L19.00003 Identifying benefits of PANS-modeling over LES for engine flows
BRANISLAV BASARA, ZORAN PAVLovic, AVL List GmbH — Large-Eddy Simulation (LES) has been frequently used for engine flows providing reliable and accurate results. On the other hand, the limitations of Reynolds-Averaged Navier-Stokes (RANS) models are very well known and some of them are possible to compensate with e.g. modelling in combustion (knock), or they are just ignored like cycle-to-cycle variability etc. Most of limitations are solved by applying LES but leading to much higher computational costs. Up to now, it has not been clear if some of hybrid RANS-LES models could improve engine calculations with a moderate increase of computational costs. The Partially-Averaged Navier-Stokes (PANS) approach (Girimaji, 2006) is designed to resolve a part of the turbulence spectrum adjusting seamlessly from RANS to DNS. In order to optimize the basic PANS method for moving geometries, Basara, Pavlovic and Girimaji (2018) introduced an additional equation for the scale supplying variable. In this approach, the main resolution parameter, the unresolved to total kinetic energy ratio, can be calculated in-situ enabling simple and cost-effective calculations of engines. Cycle-to-cycle variations have been achieved with amplitudes of cycle relevant variables that are depending on the ratio between unresolved and total kinetic energy. Nevertheless, benefits in PANS calculations are also expected to come from the proper modelling of the wall including the heat transfer even for larger \( g+ \) values as RANS models are applied there. This should have a positive impact on the emission predictions. The work presented here will try to provide answers on all these questions.

2:24PM L19.00004 Enhancement of PANS with Non-Linear Eddy Viscosity Closure
SAGAR SAROHA, KRISHNENDU CHAKRABORTY, SAWAN SUMAN SINHA, IIT Delhi, SUNIL LAKSHMIPATHY, Gecon AS, Norway — In recent years partially-averaged Navier-Stokes (PANS) method has emerged as a promising bridging method for simulating turbulent flows. However, most PANS simulations reported in the literature have been performed employing linear constitutive equation for unclosed stresses. While PANS inherently addresses the limitations of Reynolds-averaged Navier-Stokes (RANS) by reducing the overly diffusive effects by choosing appropriate sub-unity values of its filter parameters, the limitations of a linear eddy viscosity model inherited from a parent RANS model still bottleneck its performance. Indeed experimental evidence suggest that in flow past bluff bodies, there are regions where the Reynolds stress tensor is substantially misaligned with the local resolved strain rate, and a simple linear eddy viscosity assumption is not realistic. We present an enhanced version of the PANS methodology which employs a non-linear eddy viscosity model. We show that this enhancement substantially improves the prediction of various hydrodynamic as well as heat transfer statistics in flows past two representative bluff bodies: a square cylinder and a sphere. This enhanced nonlinear PANS methodology is presented as a more potent bridging method than its linear counterpart.

2:37PM L19.00005 Sensitized RANS modeling of turbulence: a scale-resolving ERM-based eddy-viscosity model
SUAD JAKIRLIC, BENJAMIN KRUMBEIN, Technische Universitaet Darmstadt, Germany, ROBERT MADUTHA, OutoTec GmbH, Oberursel, Germany, CAMERON TROPEA, Technische Universitzaet Darmstadt, Germany, 1 TEAM — A near-wall URANS (Unsteady Reynolds-Averaged Navier-Stokes) eddy-viscosity model based on elliptic-relaxation methodology (ERM) is sensitized to resolve fluctuating turbulence by introducing an appropriately formulated source term in the scale-supplying equation governing the turbulent inverse time scale. The latter term, inspired by the scaleadaptive simulation concept (Menter and Egorov, FTAc 85, 2010), enables an adequate suppression of the modeled turbulence intensity toward the respective sub-scale level. It implies the model’s self-balancing between the resolved and modeled (unresolved) contributions to the turbulence kinetic energy. The feasibility of this grid-spacing-free model formulation is checked by computing a series of internal heat and fluid flow configurations featuring boundary layer separation, flow impingement, thermal mixing of flow-crossing streams as well as flows over rough and porous walls. Comparison to under-resolved Large-Eddy Simulations (LES) applying the dynamic Smagorinsky model and to a hybrid RANS/LES model based on the same eddy-viscosity scheme, denoted by VLES (Very LES, Chang et al., IJHFF 49, 2014) indicates an advantage in terms of the predictive capabilities of the presently proposed model, especially on relatively coarser meshes.

1TRR150 project granted by the German Research Foundation
simulations of turbulent channel flow. IMU based controllers. Controllers make use of inertial measurement units (IMUs) located in the main body and require motion to occur before attempting to counter with a 1KHz sampling rate and its roll was recorded. Long short-term memory (LSTM) neural networks processed the information gathered by the acquired data. The approach for from planar experimental measurements can be fed into a turbulent channel flow DNS to reduce the required computational time and domain for a given friction Reynolds number. The effect of inlet resolution and required spanwise extent are examined by generating synthetic experimental fields from streamwise periodic channel flow DNS at Re = 180 and 550 and using this data as the inlet to a channel flow DNS with inlet-outlet boundary conditions. When fully resolved inlet data is used the streamwise domain of the inlet-outlet DNS can be reduced to 1/16 of the periodic domain with minimal influence on the flow statistics and can even withstand a halving of the spanwise domain.

Monday, November 25, 2019 1:45PM - 3:16PM
Session L20 Low-Order Modeling and Machine Learning

2:11PM L20.00003 Bringing Computational Fluid Dynamics at the heart of industrial processes: can Machine Learning help? . CHRISTOS VARSAKELIS, SANDRINE DESSOY, GlaxoSmithKline Biologicals — Advances in both physical understanding and computational power have rendered several industrial problems amenable to a Computational Fluid Dynamics (CFD) analysis. However, even though in silico prototyping is gradually becoming the norm, utilizing the predictive power of CFD for real-time tasks, e.g. controlling a process, remains prohibitive. Recent literature has suggested that Machine Learning (ML) algorithms may be trained by CFD simulations and, subsequently, replace CFD codes due to their speed advantage. In this talk, we systematically evaluate this proposal through a series of industrial test cases of varying complexity. Both laminar and turbulent test cases are examined for various spatio-temporal scales. The talk concludes with the ranking of the performance of ML algorithms in terms of accuracy, speed to completion are often limiting factors for the problems that can be studied numerically. At the same time, problem-specific extensions to the simulation code are often necessary. The Julia programming language promises to enable quick, iterative development in a friendly, high-level language while achieving a performance comparable to Fortran and C. We present a new code for direct numerical simulation of turbulent channel flows written in Julia, scaling to thousands of CPU cores. We compare its performance to a Fortran code with the same numerical approach and discuss advantages and drawbacks of the new code.

2:24PM L20.00004 Prediction of aerodynamic loads in turbulent flow conditions . ANDREAS NATSIS, Portland State University, RAL BAYON CAL TEAM, ALEXANDER J. HUNT TEAM — In the direct numerical simulation (DNS) of turbulence it is the large scales that require large computational domains and long simulation times to attain the correct statistically stationary state. In contrast experimental measurements struggle to resolve down to the dissipative flow scales, but have far less trouble capturing the large scales. In this work we demonstrate how large scale information from planar experimental measurements can be fed into a turbulent channel flow DNS to reduce the required computational time and domain for a given friction Reynolds number. The effect of inlet resolution and required spanwise extent are examined by generating synthetic experimental fields from streamwise periodic channel flow DNS at Re = 180 and 550 and using this data as the inlet to a channel flow DNS with inlet-outlet boundary conditions. When fully resolved inlet data is used the streamwise domain of the inlet-outlet DNS can be reduced to 1/16 of the periodic domain with minimal influence on the flow statistics and can even withstand a halving of the spanwise domain.

Note: The authors acknowledge the support of The Australian Research Council through a Discovery Grant and the European Research Council through the advanced grant COTURB.
2:37PM L20.00005 Airfoil control with Proximal Policy Optimization, DENIS DUMOULIN, PHILIPPE CHATELAIN, UCLouvain — Airfoil control generally relies on techniques based on a dynamical model of the actual system one wants to control. Such methods are in some cases limited by the expressiveness of the model and its linearization around equilibrium points. This is the case in aerodynamics where non-linearities (e.g., turbulent structures or dynamic stall) strongly affect the flight conditions and more specifically aerodynamic loads undergone by the flying body. To avoid such limitations, we propose the use of a model-free control method based on Reinforcement Learning (RL) algorithms. This study relies on a low-order inviscid planar solver which accounts for separation; this model controls its computational cost through lumping of the shed vortices in the wake. The complexity of the model is critical for RL methods which require large amounts of data. We then use Proximal Policy Optimization (PPO) to control the dynamics of a NACA0012 airfoil. This planar body goes through a random train of vortical dipoles while trying to keep its aerodynamic coefficients constant. Within a few thousands of episodes of training, the controller is able to stabilize the airfoil regardless of the incoming vortices using pressure measurements distributed on the body and its kinematics.

2:50PM L20.00006 A reduced order model for store separation in high speed flow, NICHOLAS PETERS, JOHN EKATERINARIS, Embry-Riddle Aeronautical University — Store separation from aircraft and spacecraft has historically been a critical issue for the aerospace industry. Given the severity of the problem much effort has been spent on the topic. Yet, the process for identifying potential failures is a resource intensive and iterative process. A potential remedy for reducing iterations in this process is the implementation of an active controller for use during separation. The objective of this study is to design such a controller, using a reduced ordered model (ROM) for the flow around an external store undergoing separation. A combination of computational fluid dynamic (CFD) solvers, fun3D and Ansys Fluent, is employed to obtain flow fields around the vehicle and the store. Preliminary validation of the numerical results is initially carried out. Representative cases of store separation are obtained next. Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD) are used to obtain leading modes that will be used to reconstruct a ROM of the flow field. These models will be compared with the objective of seeing which method better represents the numerical solution to the problem at different flow speeds with and without the presence of shock waves. The ROM will then be used to construct the controller.

3:03PM L20.00007 Fast potential flow computations for low-order aerodynamic modelling, DIEDERIK BECKERS, JEFF D. ELDREDGE, University of California, Los Angeles — Lightweight aircraft are vulnerable to flow separation induced by gusts. For purposes of regulating flight in the presence of such gusts, it is important to estimate the flow behavior and the instantaneous aerodynamic forces. In previous work, it was shown (D. Darakananda et al., Phys. Rev. Fluids 3, 124701, 2018) that low-order vortex models can be assimilated with sensor measurements to achieve this estimation. However, traditional vortex element models using Biot-Savart interactions can be computationally inefficient, particularly for 3D models. This work addresses grid-based computations for potential flows in 2D and 3D with the goal of implementing a low-order vortex model for fast modeling of separated aerodynamic flows and gust interactions. The immersed boundary projection method is used to solve for the vector potential field subject to the constraints introduced by the presence of a body. The equations are discretized on a staggered Cartesian grid and solved using the lattice Green’s function. The accuracy of these computations is demonstrated for singular vortex elements in 2D and the extension to 3D flows will be discussed.

1This work has been supported by AFOSR, under award FA9550-18-1-0440.

Monday, November 25, 2019 1:45PM - 3:16PM – Session L21 Drops: Impacts with Solids II 603 - Jeffrey Marshall, Univ Vermont

1:45PM L21.00001 Rebound and Sticking Dynamics of Droplets Impinging on Wettable Surfaces, ANUPAM MISHRA, YANBAO MA, Department of Mechanical Engineering, University of California, Merced, ARVIND GOPINATH, Department of Bioengineering, University of California, Merced — We investigate the rebound and sticking dynamics of liquid droplets impinging on wettable surfaces under zero gravity conditions using Multi-body Dissipative Particle Dynamics. A soft potential with an attractive and a repulsive component is used to model surface tension for liquid. The surface-droplet interactions are modeled by a similar potential with independently tunable attractive and repulsive components to obtain a wide range of wettability. The viscosity of the ambient medium is set to zero and the droplet density is held constant. Varying the attractive liquid-solid potential, the droplet velocity and size independently, we classify the impact dynamics into one of two categories - rebound or stick. Collapsing the results in two dimensionless parameters - the Weber number \(W_e\) based on drop properties, and an attraction parameter \(A_e\) based on the surface-drop potential - we obtain the critical curve separating the rebound from stick responses. Upon impact, both rebounding and sticking droplets form a pancake-shaped disk on the substrate. We calculate the evolution of the radius of the disk, track the center of mass distance from the substrate, calculate the rebound time and compare with analytical scalings in the literature.

1:58PM L21.00002 Bouncing Dynamics of Liquid Marbles, NIKOLAY IONKIN, DANIEL HARRIS, Brown University — Liquid marbles are millimetric droplets of fluid coated in a hydrophobic powder, which behave like soft solids that can readily roll and bounce. In this talk, we demonstrate that a liquid marble bouncing on a vertically vibrated surface demonstrates a period-doubling cascade to chaos as the vibration amplitude is increased. The resulting sequence of bifurcations is highly reminiscent to that of the extensively studied 1D model of a bouncing ball on a vibrating platform. Unlike the classical model however, our bouncer is relatively soft, and thus the time evolution of both the center of mass and the contact point is significantly affected by the vibration period. The experimental results are directly compared to predictions of a simple bouncing spring model. Our findings may help further elucidate the subtle mechanical behavior of these complex fluid objects.

2:11PM L21.00003 Deformation and bursting of elastic capsules impacting rigid walls, ETIENNE JAMON-PUILLET, TREVOR JONES, PIERRE-THOMAS BRÜN, Princeton University — From water balloons to cells, eggs and various organs, thin elastic shells enclosing a liquid core, or capsules, are ubiquitous. Yet, despite their prominence, an understanding of how capsules deform and burst under impact is lacking. Here, combining controlled experiments with formal models, we study the deformation of elastic capsules impacting rigid walls. We unravel a strong analogy with liquid drop impact; the shell surface modulus taking the place of the surface tension, albeit the details of the scalings are different. By computing the shell elastic energy during impact and performing a detailed energy balance, we rationalize this analogy and quantitatively predict the capsule maximal deformation for liquids with viscosities up to 1000 cP (Re > 100). Unlike drops however, capsules can burst and be pre-stretched. Experiments show a significant shift in the critical burst velocity induced by the pre-stretch, a feature also captured by our model once pre-stretch is included.
2:24PM L21.00004 Modeling of Volcanic Ash Impingement in Gas Turbine Engines 1
, DRUE SEKSINSKY, JEFFREY MARSHALL, University of Vermont — Impact of volcanic ash particles on heated surfaces of gas turbine engines (GTEs) are a significant concern for the aviation community. Deposition of molten volcanic ash particles within GTEs has led to aircraft engine damage and shut-down, as well as airspace closure within a wide region surrounding volcanically active areas. Unlike most studies of droplet-surface impact, in this problem the molten liquid viscosity is sufficiently large that the droplet Reynolds number (Re) is significantly less than unity. The research describes numerical simulations of low Re droplet impact using the combined level-set-volume-of-fluid (CLSVOF) method, as well as a simplified theory for the low Re impact process. Two distinct phases of the low Re droplet impact process are observed. In the first phase, the droplet kinetic energy decreases rapidly, which is roughly balanced by the energy dissipation. In the second phase, the droplet motion is controlled by potential energy balance with viscous dissipation. The second phase occurs over a time period that is much larger than that of the first phase. Since there is little experimental data for droplet collisions in this regime, we are using our numerical results to guide and validate the theory development.

1This work was supported by NASA under cooperative agreement number NNX15AK55A.

2:37PM L21.00005 Impact of a compound drop produces fine radial jetting 1, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology (KAUST), Thuwal, Saudi Arabia. JIAMING ZHANG, Physics of Fluids Group, University of Twente, The Netherlands, ERQIANG LI, University of Science and Technology of China, Hefei, China — We study the impact of a compound drop on a solid surface and the fine radial jets, which emerge at high speed near the solid substrate. The drop is made in a flow-focusing micro-fluidic device feeding smaller droplets into the pendent drop at a nozzle. The outer continuous phase consists of water-glycerin mixtures of various viscosities, while the disperse-phase inner droplets are of a much heavier perfluorohexane liquid. The inner droplets therefore sink to the bottom of the pendent drop before its release from the nozzle. We use a handful of inner droplets which can arrange into regular patterns around the axis of symmetry. The early impulsive stage of the impact forms a radial jet under each of the inner droplets, which emerge at about 8 times faster than the drop-impact velocity, but are only 40 microns thick. We use a million fps bottom imaging, through a glass substrate, to show that the jets are formed by flow-focusing under the inner droplets. We systematically change the droplets, which emerge at about 8 times faster than the drop-impact velocity, but are only 40 microns thick. We use a million fps bottom imaging, through a glass substrate, to show that the jets are formed by flow-focusing under the inner droplets. We systematically change the number of inner droplets and the viscosity of the continuous phase to identify the splashing threshold for this configuration. The inner droplets are greatly deformed and break up into smaller satellites by viscous stretching, involving capillary pinch-off or tip streaming.

1This work was funded by King Abdullah University of Science and Technology (KAUST) under grant URF/1/3727-01-01.

2:50PM L21.00006 Impact of Highly Concentrated Milk Droplets with Clean and Fouled Surfaces, MIGUEL BALZAN, F. STEVEN WELLS, GEOFF WILLMOTT, The Department of Physics, The University of Auckland — Wall deposition in milk spray dryers occurs due to the repeated impact of milk droplets against solid surfaces. Milk can be characterized as a solution of water with solids. Since there is limited information on the literature about single milk droplet collisions against solid surfaces, our study focuses on exploring this phenomenon by characterizing the effect that the surface properties and solids concentration have on the spreading dynamics. Substrates of varied nature; either clean, chemically treated with acid, and milk-product fouled surfaces, were used, resulting in mean roughness values that ranged from 0.023 to 6.270 μm. Our results indicate that the droplet dynamics are weakly dependent on the surface roughness during the kinematic and spreading stages after drop impact. The effect of solids concentration does play a significant role in the spreading and recession dynamics. The fluid behavior is Newtonian up to solids concentrations approximately equal to 20%. Afterward, the fluid is shear-thinning. We quantified the effect that increasing the solid concentration had on the maximum spreading dynamics, finding that increasing the concentration from 10% up to 40% reduced the maximum spreading diameter after impact by approximately 40%.

3:03PM L21.00007 ABSTRACT WITHDRAWN

Monday, November 25, 2019 1:45PM - 3:16PM
Session L22 Drops: Heat Transfer and Evaporation II
604 - Paul Steen, Cornell University

1:45PM L22.00001 Evaporation of sessile drops on soft membranes with capillary origami 1, YUHONG CHEN, DANIEL OREJON, PRASHANT VALLURI, VASILEIOS KOUTSOS, KHELLIL SEFIANE, the University of Edinburgh — Drops evaporating on soft substrates are common in nature, such as raindrops on tree leaves. If the substrate is thin enough, it can get deformed by the action of surface tension of a drying drop. This folding is known as ‘capillary origami’, which provides a simple and cheap method for fabricating predetermined 3D structures at the micro/nano-scale. Here, we explore the influence of capillary origami on drop evaporation. Our experiments concern with evaporation of sessile microliter water drops on soft PDMS membranes. We consider the effect of substrate thickness noting that bending stiffness is a cubic function of thickness. Thus, drying sessile droplets of the same size can lead to either absent, partial or complete folding of the substrate. The evolution of evaporation fluxes is obtained through a microbalance with a resolution of 0.1 mg/s, and is simultaneously imaged by a side-view CCD camera and a top-view optical camera, respectively. These videos are processed using ImageJ to extract the data and quantify the folding extent of the membranes. Our results show that foldable membranes reduce evaporation rates of sessile droplets thereby lengthening the drop lifetimes.

1This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (No. 2019R1A6A1A03033215).

1:58PM L22.00002 Inclination effect on evaporation of colloidal droplets 1, JIN YOUNG KIM, Research Center for Advanced Materials Technology, SSKU Advanced Institute of Nanotechnology, Sungkyunkwan University, MARTA GONCALVES, SSKU Advanced Institute of Nanotechnology, Sungkyunkwan University, HYOUNGOO KIM, Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, BYJUNG MOOK WOON, School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology, Sungkyunkwan University — When colloidal droplets including micro- and nanoparticles evaporate on flat solid substrates, commonly ring-like depositions are generated by continuous outward capillary flows. When the substrate is inclined, droplets become asymmetric and non-spherical by gravity and it affects evaporation dynamics and particle deposition. However, inclination effects on evaporation and deposition are not clear yet. Here we observe evaporation and deposition of colloidal droplets by changing particle sizes and tilting angles. We find that there is competition between downward sedimentation flows by gravity and upward capillary flows by evaporation, inducing that uniformity of coffee-rings depends on particle sizes and tilting angles. For small particles, evaporation-driven upward flows are more dominant than gravity-driven downward flows.

1This work was funded by EC-H2020-RISE-ThermaSMART-778104 and CSC.
A non-equilibrium multi-component evaporation model for blended diesel/alcohol droplets. HONGYUAN ZHANG, PING YI, SUO YANG, University of Minnesota — A non-equilibrium Langmuir-Knudsen model for multi-component pure diesel and blended diesel/alcohol droplets and sprays is developed. This model takes into account most of the key processes during the droplet lifetime, including the finite heat conduction and limited mass diffusion inside the droplet, the differential diffusion in gas phase, and the non-equilibrium Langmuir-Knudsen evaporation law for multi-component droplets. The present model shows good agreements with experimental measurements for pure ethanol, diesel, and blended diesel/ethanol droplets. The non-equilibrium effects become significant when the initial droplet diameter is smaller than 20 μm, and these effects are enhanced with increasing ambient temperature and forced convection intensity. The non-equilibrium effects are more significant for the blended diesel/alcohol droplets than pure diesel, especially during the evaporation period of ethanol. The present evaporation model has also been applied to calculate the evaporation processes of single- and multi-component fuel sprays under various ambient conditions. The non-equilibrium effects for the blended diesel/alcohol sprays are significant in terms of the fuel vapor component concentrations.

Channel Flow with Evaporation of Spatially Resolved Droplets GIANDOMENICO LUPO, KTH, ANDREA GRUBER, SINTEF, CHRISTOPHE DUWIG, KTH — We perform direct numerical simulations (DNS) of more than 14000 cold spherical droplets evaporating in hot turbulent channel flow ($Re_\tau = 180$, 5% initial liquid volume fraction), at conditions representative of industrial applications. Four-way coupling of the droplet motion with the turbulent carrier phase and interface-resolved evaporation dynamics allow us to fully describe the intra- and inter-phase exchange of heat, species and momentum transfer, at all relevant scales of motion. We analyze the modulation of turbulence by the dispersed phase, and the associated migration of the droplets towards the channel centreline. The redistribution of the droplets has a strong impact on the evaporation dynamics, which we characterize by the joint probability density function of the evaporation rate and the droplet distance from the wall. This leads to an inquiry of the feasibility of film theory for modelling the present evaporation regime, and the definition of an appropriate nondimensional parameter for the scaling of the evaporation rate (correlation of Sherwood number). Finally, we investigate the influence of the energy boundary condition (adiabatic vs. isothermal walls) on the evaporation dynamics.

Experimental and numerical study of the flow instabilities during sessile droplet evaporation in microgravity. SANJEEV KUMAR, MARC MEDALE, DAVID BRUTIN, Aix-Marseille University, JUSTI TEAM — Our current research is focused on thermal Marangoni instabilities in sessile ethanol and Hydronfluoroether (HFE7100) droplets, which develop spontaneously during forced evaporation. We will present the numerical modelling in 3D unsteady with moving interface of a sessile droplet under forced evaporation and showing internal flow instabilities. We assumed a pinned contact line and a spherical-cap shape of the liquid-gas interface. Our computations contribute to figure out the internal 3D flow structure in the droplet and also to determine the driving mechanism and energy sources of the observed thermo-convective instability and thus clarifies its nature. We will compare the numerical results with experimental results obtained with ARLES experiment in microgravity during the ESA SSC MASER-14 rocket campaign for the HFE7100 droplets and the parabolic flight campaign CNES VP139 and ESA VP140 for the ethanol, methanol and pentane droplets.

The role of substrate geometry on droplet evaporation. JOHN MCCARTHY, Department of Engineering Science, University of Oxford, DOMINIC VELLA, Mathematical Institute, University of Oxford, ALFONSO A. CASTREJON-PITA, Department of Engineering Science, University of Oxford — The coffee ring effect is the name given to the phenomenon where particles suspended in an evaporating droplet migrate towards the contact line, producing characteristic dense rings of particles. It is observed in many everyday and technological settings, from stains after coffee spills to imperfections in inkjet printed objects. Methods to control and even suppress this - often undesired - effect include the use of tailored mixtures of solvents to control evaporation, the use of elongated particles, the use of electrowetting and surface acoustic waves to induce internal flows, etc. All these methods have the objective of reducing, eliminating or overcoming the capillary flows leading to the deposition of suspended particles along the contact line during (differential) evaporation. Each method limits to some extent the type of droplets for which it can be used. We explore the role played by substrate geometry in controlling the evaporation of droplets and its role on the formation, or otherwise, of a coffee ring.

Effects of contact lines on evaporative heat transfer of liquid films JUNGTAEK KIM, HANSEUL Choi, YUN SEOG LEE, HO-YOUNG KIM, Seoul National University — Falling film evaporators cool liquids that flow through a bundle of tubes, over which refrigerants are sprayed and evaporated. A key parameter to determine the heat transfer coefficient is the film Reynolds number ($Re_f$) that depends on the spraying amount of refrigerant. Because dry-out of tube walls occurs when $Re$ is too low while the liquid film becomes too thick for too high $Re$, an optimal $Re$ exists that maximizes the heat transfer coefficient. Here we show that the optimal $Re$ for evaporative heat transfer over geometrically patterned tube walls is also critically affected by the enhanced mass transfer rate at the solid-liquid-gas three phase contact lines. It is experimentally found that the heat transfer of partially filled grooves with contact lines is more efficient than completed wet grooves, despite the decreased heat transfer area. We experimentally measure the heat transfer coefficient from various surface patterns under contact line effects. Then we provide a mathematical model to optimize enhanced surface features maximizing evaporative heat transfer coefficient.

Monday, November 25, 2019 1:45PM - 3:16PM —
Session L23 General Fluid Dynamics: Mathematical Methods 605 - Tom Mullin, Oxford

1:58PM L23.00002 An extension to the extended Beltrami solution method and solutions to the 3D Navier-Stokes equations1. NOLAN DYCK, ANTHONY STRAATMAN, The University of Western Ontario — Recirculating flows, such as those seen in vortex separators, swirl combustion chambers, and Ranque-Hilsch vortex tubes can seldom be modelled analytically, and where exact solutions may be found, viscous effects are neglected. In this work we attempt to describe swirling motions, by extending the extended Beltrami solution method for the Navier-Stokes equations to describe 3D flows where the flow variables may be written as a function of two spatial co-ordinates. The new formulation emerges when auxiliary functions are added to the general Bragg-Hawthorne equations and reinserted into the governing equations. As is the case with the extended Beltrami approach, solutions are sought by guessing forms of the auxiliary functions, and attempting to solve the system of equations. Using this technique, we find many new solutions. Well known planar flows including Kovasznay flow and Wang’s flow have been generalized to three-dimensions. A unique solution in plane polar co-ordinates has been found. A new 3D swirling flow solution which can be calculated the angular analogue to Kovasznay flow is found which exhibits many realistic features observed in swirling flow applications.

The authors gratefully acknowledge the financial support from the Natural Sciences and Engineering Research Council of Canada (NSERC).

2:11PM L23.00003 Exact Solutions for Flows in Periodic Domains. PETER BADDOO, DARREN CROWDY, Imperial College London — In this work it is shown that recent advances in conformal geometry can be leveraged to obtain analytic solutions for flows in periodic domains. The solutions are constructed in a parametric circular domain and then conformally mapped to the periodic physical domain. By expressing the problem in terms of the transcendental Schottky–Klein prime function, the ensuing solutions are valid for domains of arbitrary connectivity, i.e. any number of objects per period window. Moreover, the conformal mapping to the desired physical domain may be constructed using a new periodic Schwarz–Christoffel formula. The mathematical analysis is valid for conformally invariant equations and is therefore applicable to a range of scenarios in fluid mechanics including potential flows, advection-diffusion problems and interfacial dynamics. Accordingly, the solutions find relevance in areas such as vortex dynamics, transport phenomena, turbomachinery flows, mesh generation and fractal growth.

2:24PM L23.00004 Comparison of Deep Learning strategies for flow field reconstruction from wall measurements. SAMIR BENEDDINE, ONERA, 8 rue des Vertugadins, FR-92190 Meudon, FRANCE — This work explores deep learning reconstruction techniques to estimate the pressure field in the laminar wake of a bluff body from a few wall measurements. It aims at comparing two main strategies: a classical approach, where a neural network maps the input (the point-wise pressure measurements) to the target (the full pressure field) by minimizing the $L_2$-norm of the error between the output of the network and ground truth, a so-called decoupled approach, based on Generative Adversarial Networks. For the latter, a network is first trained to generate fields that are not distinguishable from actual pressure field obtained from a simulation. Then, given input measurements, it searches for the best match among all the possible field that it can generate. This generative strategy is a state-of-the-art approach for image inpainting, and it presents several strong advantages over the first method. For instance, it is totally flexible with respect to the input measurements: the same network can carry out the reconstruction no matter the location or the number of input sensors. Other interesting advantages will be detailed during the presentation, such as the ability of generative approaches to be used for temporal super-resolution.

2:37PM L23.00005 Deep Learning Time-Dependent Hagen-Poiseuille Flow. NINA PRAKASH, WEI HU, AMIR BARATI FARIMANI, Carnegie Mellon University — Deep learning has emerged as a valuable tool for data-driven modeling of fluid flow. Rather than a traditional physics-based computational approach based on solving equations, deep learning can be used to derive models of flow systems directly from experimental or simulation data. In this work, we use Long Short Term Memory (LSTM) to learn the velocity profile development of Hagen-Poiseuille Flow in its time-dependent entry region. Given arbitrary boundary conditions including the pipe diameter, fluid properties, and axial pressure gradient, the network is able to predict the entry region velocity profiles as flow develops from rest with a mean squared error of less than 1%. The model is trained solely on simulation data and thus is able to learn the behavior of the system with no knowledge of the physical mathematical model. This work contributes to a growing body of work on the application of machine learning to fluid modeling by proving the success of LSTM to model a time-dependent fluid flow system. The deep learning framework presented in this work has the potential to be successful for systems that involve complex geometries or turbulence that can make traditional approaches computationally inefficient and cumbersome, or for systems for which the underlying mathematical model is unknown.

2:50PM L23.00006 On the Advection-Diffusive Mass Transport of Gas Mixtures. ALEX JARAUTA, VALENTIN ZINGAN, PETER MINEV, MARC SECANELL, University of Alberta, DEPARTMENT OF MATHEMATICAL AND STATISTICAL SCIENCES, UNIVERSITY OF ALBERTA TEAM, ENERGY SYSTEMS DESIGN LABORATORY, DEPARTMENT OF MECHANICAL ENGINEERING, UNIVERSITY OF ALBERTA TEAM — Mass transport of gas mixtures often occurs in a variety of engineering applications such as fuel cells and coal fired power plants. Classical convection-diffusion equations are limited to binary mixtures and dilute species in a mixture. Also, these theories have been shown to be unable to reproduce several phenomena occurring in capillaries or small pores [1], such as osmotic diffusion (i.e., diffusion without a concentration gradient), reverse diffusion (i.e., diffusion in the direction of a positive concentration gradient), and diffusion barrier (i.e., no diffusion with a concentration gradient). The limitations of these classic models stem from the fact that only a mass-averaged velocity field is considered. In this work, a new multiphase mass transport model was developed based on the work of Kerkhof and Geboers [1]. This model considered the velocity of each individual species, as well as an individual momentum equation. The Stefan tube diffusion experiment was used to compare our model to the advection-diffusion equation. Partial viscosities and gradients of species velocities were identified as key parameters to overcome the limitations of the advection-diffusion equation. References: [1] P.J.A.M. Kerkhof and M.A.M. Geboers, AIChE J., 51(1):79-12.
3:03PM L23.00007 On the structure of distribution function in kinetic methods and its implications, LIAN-PING WANG, Southern U of Sci and Tech and U of Delaware — This theoretical talk will provide a derivation of the structure of the distribution functions in kinetic schemes such as the lattice Boltzmann method and the discrete unified gas kinetic scheme. Starting from the continuous Boltzmann equation with the BGK collision model and an external force field, the structure of the distribution functions, in terms of the macroscopic variables, is derived by using only the Chapman-Enskog expansion for continuum flows. The result then is used to understand why a source term in the kinetic equation can be designed to adjust fluid viscosity and thermal conductivity, etc. The structure of the discrete distribution functions used in the kinetic methods is then derived by proper transformation and the use of Gauss-Hermite quadrature. The result then provides a basic framework to discuss proper implementation of boundary condition in the kinetic methods. Previous boundary implementation methods will be examined under this framework and alternative boundary implementation methods will be explored. Possibility of using this approach to explore truncation errors in kinetic methods will also be discussed.

Monday, November 25, 2019 1:45PM - 3:16PM
Session L24 Bubbles: Growth, Heat Transfer and Boiling II

1:45PM L24.00001 ABSTRACT WITHDRAWN

1:58PM L24.00002 Light-guided surface plasmonic bubble movement via contact line de-pinning by in-situ deposited plasmonic nanoparticle heating.1. QIUSHI ZHANG, EUNGYU LEE, YUNSONG PANG, Department of Aerospace and Mechanical Engineering, University of Notre Dame, 365 Fitzpatrick Hall, Notre Dame, IN 46556, JARROD SCHIFFBAUER, Department of Physics, Colorado Mesa University, Grand Junction, CO, USA, ALEKSANDAR JEMCOV, HSUEH-CHIA CHANG, TENGFEI LUO, Department of Aerospace and Mechanical Engineering, University of Notre Dame, 365 Fitzpatrick Hall, Notre Dame, IN 46556 — Precise spatio-temporal control of surface bubble movement can benefit a wide range of applications like high-throughput drug screening, combinatorial material development, microfluidic logic, colloidal and molecular assembly, etc. In this work, we demonstrate that surface bubbles on a solid surface are directed by a laser to move at high speeds (>1.8 mm/s), and we elucidate the mechanism to be the de-pinning of the three-phase contact line (TPCL) by rapid plasmonic heating of nanoparticles (NPs) deposited in-situ during bubble movement. Based on our observations, we deduce a stick-slip mechanism based on asymmetric fore-aft plasmonic heating: local evaporation at the front TPCL due to plasmonic heating de-pins and extends the front TPCL, followed by the advancement of the trailing TPCL to resume a spherical bubble shape to minimize surface energy. The continuous TPCL drying during bubble movement also enables well-defined contact line deposition of NP clusters along the moving path. Our finding is beneficial to various microfluidics and pattern writing applications.

1This work is supported by National Science Foundation (1706039) and the Center for the Advancement of Science in Space (GA-2018-268).

2:11PM L24.00003 The physics of plasmonic vapor-gas bubbles in a gassy liquid, YUHANG ZHANG, Johns Hopkins University, ANDREA PROSPERETTI, University of Houston — When illuminated by resonant irradiation of a continuous-wave laser, gold nanoparticles deposited on a surface immersed in a liquid generate huge amount of heat in a very short period of time, leading to the nucleation of vapor bubbles referred to as plasmonic bubbles. In this work, a spherically symmetric mathematical model is proposed to describe the various physical processes that affect the dynamics of these bubbles: growth, condensation, the diffusion of dissolved gas into and out of the bubble and the attendant mass and heat transfer. The model is solved by transforming the partial differential equations into a system of ordinary differential equations using a collocation method. The different phases of the bubble behavior on short (microseconds) and longer (milliseconds to seconds) time scales found in experiments are reproduced by the numerical simulations. The effects of the degree of dissolved gas saturation in the liquid are discussed.

2:24PM L24.00004 Direct Numerical Simulations of Surfactant Effects on Heat and/or Mass Transfer Around a Bubble1. THOMAS ABADIE, OMAR MATAR, Imperial College London — Contaminants or surfactants are involved in a wide range of environmental and industrial applications and their presence can affect significantly both the dynamics and transfer phenomena around bubbles and droplets. In this study, a front-tracking method is presented and assessed in order to model accurately surface tension forces for capillary driven flows and heat and mass transfer around a bubble rising in a continuous liquid phase. The effects of Marangoni stresses on the bubble dynamics on the one hand and on the Sherwood or Nusselt number on the other are investigated as a first step towards improve the understanding of heat and/or mass transfer in contaminated bubble swarms.

1PETRONAS and Royal Academy of Engineering Research Chair in Multiphase Fluid Dynamics for OKM; RCS (Research Computing Service of Imperial College London) for computational resources.

2:37PM L24.00005 Bubble Dynamics on Nanostructured Microwires1. LAUREN COERTZE, DANIEL OREJON MANTECON, MARILIZE EVERTS, PRASHANT VALLURI, JOSUA MEYER, KHELLIL SEFIANE — Boiling on microwires is investigated aiming to provide a better understanding of the bubble dynamics and heat transfer as these are of great importance to many industrial and everyday processes. It is proposed that nanoparticle surface coatings may be a simple and scalable method of modifying the surface wettability and structure with the associated differences in bubble-surface interactions and the consequent variations in bubble dynamics, critical heat flux (CHF) and heat transfer coefficient. This work aims to develop an improved understanding of bubble dynamics such as bubble velocity, growth rate, bubble density distribution and detachment frequency on coated microwires at various heat fluxes. Experimental investigations will consider nanoparticle coated and bare platinum microwires with diameters of 100 and 250 micrometres, in pool boiling with water as working fluid. High speed, high resolution videography will be used to observe bubbles from nucleation to departure. The analysis will focus on the bubble dynamics occurring on nanoparticle coated and bare microwires. Bubble dynamics and CHF for the coated surfaces are expected to change compared to the bare wire depending on the nature of the surface coating applied.

1The authors gratefully acknowledge ThermaSmart for the support.
2:50PM L24.00006 Heat transport by bubbles in vertical natural convection\textsuperscript{1}, CHONG SHEN NG, University of Twente, ROBERTO VERZICCO, University of Rome Tor Vergata, DETLEF LOHSE, University of Twente — We consider a basic configuration of bubbles in vertical natural convection. The datasets are obtained from direct numerical simulations for one decade of Rayleigh numbers, a Prandtl number of 7 and the bubbles are simulated with immersed boundaries using the interaction potential approach. By separately enabling the thermal and mechanical coupling, we show evidence that the heat transport is enhanced when the bubbles are both thermally and mechanically coupled to the flow. When only pure mechanical coupling is considered, we instead find a lower heat flux in the system. The enhanced heat flux from the addition of thermal transport highlights the importance of thermal transport by bubbles in this setup. To shed light on the details of the mechanism, we discuss the contributions to the heat flux with reference to local statistics of the thermal boundary layers.

\textsuperscript{1}This work is part of the research programme of the Foundation for Fundamental Research on Matter (FOM) with project number 16DDS001, which is financially supported by the Netherlands Organisation for Scientific Research (NWO).

3:03PM L24.00007 Direct numerical simulation of heat transfer in turbulent bubbly pipe flow\textsuperscript{1}, IN-KOO LEE, JAEHEE CHANG, HAECHEON CHOI, Seoul National University — In pipe flows occurred in a reactor of a nuclear power plant and a radiator in a car, forced convection with bubbles occasionally occurs in an undesirable manner. These bubbles significantly change the flow structures and heat transfer in a pipe. We perform direct numerical simulation of fully developed turbulent bubbly flow with heat transfer in a vertical pipe to examine the variations of flow structure and heat transfer due to bubbles. The phase interface is tracked by level-set method in the Cartesian coordinates. The simulation results show that heat transfer is enhanced by the motion induced by counter-rotating vortices existing in the rear of the bubble. As the bubble volume fraction increases, the radial distribution of bubbles becomes flat due to the interaction among bubbles. This flat bubble distribution results in flattened temperature profile in the radial direction. Therefore, the rate of increase in the heat transfer coefficient decreases with increasing bubble volume fraction.

\textsuperscript{1}This work was supported by the NRF(NRF- 2017M2A8A4018482) programs of MSIP.

Monday, November 25, 2019 1:45PM - 3:16PM – Session L25 Vortex Dynamics and Vortex Flow: Theory

1:45PM L25.00001 The motion of buoyant point vortices\textsuperscript{1}, ANIRBAN GUHA, School of Science and Engineering, University of Dundee, JEFF CARPENTER, Institute of Coastal Research, Helmholtz Zentrum Geesthacht — A general formulation is presented for studying the motion of buoyant vortices. It extends the well-known Hamiltonian framework for interacting homogeneous point vortices to include buoyancy effects acting on the vortices. This is then used to systematically examine the buoyant 1-, 2-, and 3-vortex problems. In doing so we find that 2 buoyant vortices may either evolve as a pair in bounded circular orbits, or as two independent unbounded vortices that drift apart, and a criteria is found to distinguish these cases. Special attention is given to the buoyant vortex couple, consisting of two vortices of equal and opposite circulation, and equal buoyancy anomaly. We show that a theoretical maximum height is generally possible for the rise (or fall) of such couples against buoyancy forces. Finally, the possibility and onset of chaotic motions in the buoyant 3-vortex problem is addressed. In contrast to the homogeneous 3-vortex problem, the buoyant vortex system shows evidence that chaos is present. We also demonstrate the chaotic advection of tracer parcels arising from the flow field induced by just 2 buoyant vortices.

\textsuperscript{1}Alexander von Humboldt foundation

1:58PM L25.00002 Performance of Overset Mesh in Modeling Generic Wakes for Underwater Swimming: SUYASH VERMA, ARMAN HEMMATI, University of Alberta — The wakes of generic stationary, and oscillating square panels are examined using Overset Grid Assembly (OGA), implemented in OpenFOAM. Using Direct Numerical Simulation (DNS), this study focused on the accuracy and effectiveness of OGA in simulating wake, through combination of multiple meshes for domain and bodies respectively. First, the wake of a stationary square panel is considered at angles of attack between 0° and 60°. The results are compared with the numerical study of Taïra et al. (2009) based on Immersed Boundary Method. Then, the wake of a pitching panel is examined at Re=2000 and St=0.2. Comparing the results with previous numerical (Senturk et al. 2018), and experimental (Buchholz, et.al. 2006) studies, show that the mean drag coefficient are in good agreement, and wake features were captured accurately. This provides sufficient evidence for the high capabilities of overset methodology in analyzing wake physics of sharp-edge bodies in oscillatory, and mixed motion. Future studies on complex shaped bodies are examined by combining fluid structure interaction with OGA, which also enables studying the effect of flexibility on hydrodynamic performance of propulsors.

2:11PM L25.00003 Evolution of Vortex Dipole Disturbances in Channel Flow, ANTHONY LEONARD, Graduate Aerospace Laboratories, Caltech — The initial-value problem for the linearized Navier-Stokes equations for channel flow with mean velocity and mean vorticity given by $U = (U(y), 0, 0)$ and $\Omega = (0, 0, -\partial U(y)/\partial y)$ is investigated for initial disturbances consisting of vortex dipoles. Approximate analytical solutions to the resulting Orr-Sommerfeld (OS)/Squire equations are found in terms of special functions, thus avoiding the need of large-scale computation. Under the assumption of linearity, the initial vortex dipole merely travels downstream with the local mean velocity $U(y)$ as a diffusive scalar. However, additional disturbances are produced continually because the quantity $U''(y)\partial y/\partial z$ appears in the OS equation as a source for transported quantity $\nabla^2 v$. In turn, $-U''(y)\partial y/\partial z$ is the source term in the Squire equation for the transported quantity $\omega_y$. We find it particularly effective to represent this additional $v$ field as an expansion in terms of eigenfunctions of the OS equation with complex values of $k_\perp$ ($k_\perp^2 = k_x^2 + k_z^2$). For the inviscid case, e.g., these solutions satisfy

$$\frac{d^2 v_j}{dy^2} - \frac{U''(y)v_j}{U(y)} = c k_{\perp j} v_j$$

(1)
2:24PM L25.00004 Tomo-PIV measurements of tethered sphere VIV onset while crossing the Hopf bifurcation\(^1\) . RENE VAN HOUT, LIOR ESHBAL, DANIEL KOVALEV, DAVID GREENBLATT, Technion—Israel Institute of Technology — Here, for the first time we examine the transient evolution of the vortical structures in the wake of a tethered sphere as the uniform upstream velocity is slowly increased and the tethered sphere (diameter \(D\)) dynamics passes through the Hopf bifurcation. Tomo-PIV experiments (at 15Hz) were performed in a closed-loop water tunnel. The volume of interest (4.5 \(\times\) 6.5 \(\times\) 2.5\(D^3\)) was located immediately downstream of the sphere. The reduced velocity, \(U^* = U/(f_N D)\), was stepwise raised from 2.2 to 4.5 in 100s \((dU/dt = 4.36 \times 10^{-4}\text{m/s})\), where \(U\) is the free-stream velocity, and \(f_N\) is the natural frequency of the tethered sphere. Large coherent structures resembling two streamwise oriented, longitudinal “legs” were observed at \(U^* = 2.5\) while the sphere was stationary. Approaching \(U^* = 3\), hairpin vortices, having a vertical plane of symmetry, were shed, alongside weaker induced hairpins that strengthened with increasing \(U^*\). While vertical symmetry was sporadically disturbed, the onset of VIV occurred around \(U^* = 3.7\), peaking at \(U^* = 4.4\), as the plane of symmetry was shifted to the horizontal.

\(^1\) Funded by the Israel Science Foundation (ISF), grant no. 1596/14

2:37PM L25.00005 Compressible swirling flow states in diverging or contracting pipes . YUXIN ZHANG, Washington State Univ., NOAH CYR, ZVI RUSAK, Rensselaer Polytechnic Institute, SHIXIAO WANG, Auckland Univ., New Zealand — The dynamics of inviscid, compressible and axisymmetric swirling flow of a perfect gas in diverging or contracting circular pipes is studied by global analysis techniques and numerical simulations. Inlet flow is described by profiles of circumferential and axial velocities and temperature with a fixed azimuthal vorticity. Outlet flow is a non-reflective zero radial-velocity state. We first solve the column flow ODE problem developed by Rusak et al. (2015) for the outlet state as a function of inlet swirl ratio, Mach number and pipe geometry. Several steady states are possible including centerline decelerated velocity states, centerline accelerated velocity, vortex breakdown states and wall-separation states. Numerical simulations using the unsteady and axisymmetric Euler equations are also conducted. They are based on Steger & Warming (1979) flux-splitting, finite-difference method. Simulations shed light on the stability of the various steady states and their domain of attraction in terms of initial conditions. Results show that increasing inlet Mach number with a fixed geometry delays the appearance of vortex breakdown and wall-separation to higher swirl levels. Pipe divergence at a fixed Mach number promotes breakdown while pipe contraction induces wall-separation.

2:50PM L25.00006 New exact solutions of the Euler equation: hybrid equilibria of Stuart vortices and point vortices . VIKAS KRISHNAMURTHY, MILES WHEELER, University of Vienna, DARREN CROWDY, Imperial College London, ADRIAN CONSTANTIN, University of Vienna — We present a large class of exact solutions to the planar, steady, incompressible Euler equation. These solutions combine the celebrated Stuart vortices with point vortices to form stationary ‘hybrid’ equilibria. These equilibria consist of a set of point vortices otherwise surrounded by a sea of everywhere smooth Stuart vorticity. The solutions can be deformed continuously and non-trivially by varying a parameter which appears as a simple integration constant in the theory. Various limits of these solutions result in purely point vortex equilibria in otherwise irrotational flow. It is also possible to construct an infinite sequence of such solutions, with increasing numbers of point vortices. In this talk, we will present several examples as well as a brief outline of the general theory.

3:03PM L25.00007 The three-dimensional stability of Lamb-Oseen vortex flows in a finite-length pipe . WEIMIN YUAN, SHIXIAO WANG, Auckland University, ZVI RUSAK, Rensselaer Polytechnic Institute — The 3D viscous flow instability modes that appear on a Lamb-Oseen vortex flow in a finite-length straight, circular pipe are analyzed. This study extends the previous stability analysis of a solid-body rotation flow. Neutral stability lines of the axisymmetric \((m = 0)\) and spiral \((m = 1)\) modes are presented in a Reynolds number \((Re)\) versus inlet swirl ratio \((\omega)\) operational diagrams for various vortex core sizes of Lamb-Oseen vortex flow. The analysis reveals the significant role of the vortex core size on the onset of the dominant flow instability. The vortex is dominated by \(m = 1\) (spiral) modes for a relatively large vortex core size. This behavior shifts to dominant \(m = 0\) (axisymmetric) modes for smaller vortex core sizes. The Reynolds-Orr equation is then used to analyze the various production terms of the perturbation’s kinetic energy in the vortex core as well as on the pipe boundaries. It is found that for a medium or small core size of the vortex the base flow in the core is actively involved with the shedding of the shed vortices. These mechanisms are mitigated when the after body is removed, such as for the flat plate.

Monday, November 25, 2019 1:45PM - 3:16PM – Session L26 Flow Control: Jets 608 - Robert Martinuzzi, University of Calgary

1:45PM L26.00001 Influence of the obstacle after-body on actuation effectiveness for wake control\(^1\) . ROBERT MARTINUZZI, ALEX LI, University of Calgary — The dynamic response of the turbulent wake to leading-edge synthetic jet actuation was investigated experimentally for a two-dimensional normal flat plate and square cylinder. The obstacles are immersed in a uniform stream of turbulent intensity less than 0.5%. The actuators are placed directly behind one of the obstacle leading edges. The leeward surface pressure and wake velocity (PIV) measurements are synchronized. For actuation frequencies approaching those of the Kelvin—Helmholtz shear layer instability, mean drag and wake velocity fluctuation levels can be reduced for both geometries. The flat plate wake is generally insensitive to lower frequency actuation. In sharp contrast, for the square cylinder the vortex shedding frequency can lock-on to subharmonics of the actuation frequency. Most striking is that the wake can be manipulated to increase or decrease drag. It will be shown that two different mechanisms underlie the low-frequency response. First, actuation results in a pulsed vortex pair which can interfere with the shear layer to trigger shedding. Second, the after-body can act to alter the vorticity flux along the shear layers and thus modify the strength of the shed vortices. These mechanisms are mitigated when the after body is removed, such as for the flat plate.

\(^1\) Natural Sciences and Engineering Research Council of Canada
1:58PM L26.00002 The effects of geometrical modifications on flow characteristics of a sweeping jet actuator.\textsuperscript{1} ABDUL RAOOF TAJIK, BARTOSZ SLUPSKI, VLADIMIR PAREZANOVIC, Khalifa University, UAE — This work studies the effects of modifications of the internal geometry of a sweeping jet (SWJ) actuator, such as the feedback channel diameter ($D_1$) and the size of its exit nozzle ($D_2$). The effects on the jet frequency and the overall pressure drop in the actuator are investigated using time-resolved flow fields from 2D-URANS simulations and local pressure data from experimental measurements. Similarly to \textsuperscript{1}, it is observed that the variation of the feedback channel diameter $D_1$ has little impact on the sweeping jet frequency magnitude. However, the spectrum of the sweeping jet yields a much better definition of the frequency peaks for larger $D_1$ diameters of the feedback channel. This is a surprising result, considering that the enlarged diameter could be expected to diffuse the feedback flow momentum and thus reduce the quality of the peak definition. On the other hand, the reduction of diameter $D_2$ of the feedback channel exit yields up to 50% increase in the SWJ oscillation frequency for the same pressure drop along the actuator axis. This could be a significant optimization of the SWJ geometry, given that reduced energy investment is required to produce the same oscillation frequency.

\textsuperscript{1}[1], J.H. Seo, et al, AIAA Journal 56(6) 2208-2210 (2018). This work has been supported by the Khalifa University of Science and Technology under Award No(s), FSU-2018-21 and CIRA-2019-025.

2:11PM L26.00003 Computational Analysis of Turbulence and Interactions in an Array of Fluidic Oscillators for Flow Control\textsuperscript{1} N. KOUKPAIZAN, D. HEATHCOTE, C.J. PETERSON, B. VUKASINOVIC, A. GLEZER, M. SMITH, Georgia Institute of Technology — The interactions between a spanwise array of fluidic oscillating jets and a separation bubble that is formed over a 2-D curved surface modeling the suction surface of a VR-12 airfoil are investigated numerically to capture the suppression of the separation with increasing actuation momentum. The high-fidelity simulations first examine the fully-resolved spatio-temporal evolution of the oscillating jets within the actuator cavity in a quiescent ambient. These findings are used to develop a boundary condition model for the actuator array in the cross flow. It is shown that the boundary condition model developed including turbulent variables provides good agreement with the experimental measurements of the flow control effects at a significantly reduced computational cost, and thereby enables additional insight into the flow physics in the vicinity of the actuators exit planes where the measurement techniques are limited by adequate access or resolution. The effect of turbulence on the interactions of the oscillating jets with the separated outer flow are also assessed.

\textsuperscript{1}Supported by the Vertical Lift Research Center of Excellence (VLRCOE).

2:24PM L26.00004 Active control of large-scale structures in turbulent boundary layers using wall-normal jets ZHUOSHUN RUAN, Department of Mechanical Engineering, The University of Melbourne, Melbourne, Australia, CHARITHA DE SILVA, School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney, Australia, IVAN MARUSIC, NICHOLAS HUTCHINS, Department of Mechanical Engineering, The University of Melbourne, Melbourne, Australia — This experimental study uses a spanwise array of nine wall-normal jets to actively perturb large-scale structures in a high Reynolds number turbulent boundary layer. The footprints of passing large-scale structures are sensed by an upstream spanwise array of hot-film sensors, which provides the input to the real-time controller. Various real-time large-scale control strategies can be tested with this facility. Downstream of the jet array, a further sensing array probes the efficacy of the control strategy. A large field of view streamwise / wall-normal plane PIV experiment also provides additional opportunities to investigate the interaction between the jet actuators and the large-scale structures. Results demonstrate that the degree of drag reduction attained is strongly correlated to how effectively large-scale positive fluctuations have been targeted by the control scheme. In addition, there is a strong correlation between the reduction of large-scale energy and the observed drag reduction. Such control effects persist to 23 (approximately 28000 wall units) downstream of the control input. Further analysis of the near-wall small scale turbulence reveals that the near-wall cycle has been modified due to the altered footprint from the controlled large-scale events.

2:37PM L26.00005 Vortex ring bifurcation in a moderate aspect ratio, rectangular orifice synthetic jet\textsuperscript{1} JOSEPH STRACCIA, JOHN FARNSWORTH, University of Colorado, Boulder — Vortex ring dynamics play an important role in setting the shape, entrainment rate, and near field unsteadiness in synthetic jets. In this study multi-planar stereo particle image velocimetry (SPIV) data obtained for an AR=13 rectangular orifice synthetic jet is used to volumetrically reconstruct the coherent structures of the jet. Analysis of this data reveals the axis switching dynamics of the primary vortex ring in addition to identifying several types of related secondary structures. Most significant however is the confirmation that the vortex rings undergo a viscous interaction called vorticity reconnection which causes the vortex rings, and the momentum of the jet, to bifurcate. The structure of the vortex ring during and after the vorticity reconnection event is reported at different phases of the jet and compared with studies of isolated vortex rings. To understand how this phenomenon is affected by the unique environment inside of the synthetic jet the Reynolds and Strouhal numbers of the jet were varied independently and centerline SPIV data was examined. From these results conclusions are drawn regarding which conditions allow for more complete vortex ring bifurcation.

\textsuperscript{1}This material is based upon work supported by the National Science Foundation Graduate Research Fellowship Program under Grant No. (DGE 1144083).

2:50PM L26.00006 Synthetic jet-based control of wing tip vortices CARLO SALVATORE GRECO, ANDREA PICCOLO, MIRKO ZACCARA, GÉRARD PAOLILLO, TOMMASO ASTARITA, GENNARO CARDONE, Università degli Studi di Napoli “Federico II” — Wing tip vortices are coherent structures generated by the roll-up process developing in the wake of a finite lifting wing. Currently, winglets are the unique passive control device employed in the general aviation to weaken these undesired vortices, reducing their related problems, although active control devices (ACD) are studied in the literature. ACD try to weaken the wing tip vortices by triggering their inherent instabilities. Although these instabilities are caused by the mutual induction of the vortices, all the experimental works in this field employ only half-wing models and report only time-averaged results. The purpose of the present experiment is testing the effects of rectangular synthetic jets, placed at the wing tips and actuated at the wing-tip vortex instability frequencies, on the wing tip vortices generated by a finite-span airfoil placed at the exit of an open jet wind tunnel. Phase-locked Stereo-PIV measurements of the flow field in the wing extended near-wake are carried out. The main observed effects of the control are the reduction of the wing tip vortex vorticity and induced rotational velocity.
3:03PM L26.00007 Constraints of scaling synthetic jet trajectories in crossflow , GIRISH JANKEE, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Synthetic jet actuators remain coveted components in flow control applications as the convection of vortex rings allows distribution of momentum in a boundary layer although the net mass flux remains zero. The ability to predict the trajectory of these vortex rings is critical for efficient and targeted usage of such actuators. In this investigation, a synthetic jet is issued into a crossflow from rectangular orifices with aspect ratios 3, 6 and 12 over a range of operating frequencies and blowing ratios, with the flowfield being captured through PIV measurements. An assessment of the trajectories culminates in scaling characteristics which encapsulate the aspect ratio, momentum ratio and the Strouhal number. We observe synthetic jets to follow identical trajectories provided the total momentum ratio between the jet and the crossflow remain the same, irrespective of the actuation frequency. However, the universality of such scaling is subject to certain constraints with a lower bound defined by the jet formation criterion and an upper bound related to successive vortex pair interactions.

Monday, November 25, 2019 1:45PM - 3:03PM — Session L27 Biological Fluid Dynamics: Insect Flight III - Small Insects

1:45PM L27.00001 Fluid-Structure Interactions in Bristled Insect Wings , MATTEO PEZZULLA, FRANCOIS GALLAIRE, PEDRO REIS,cole polytechnique fédrale de Lausanne — The wings of some small insects, such as the Mymaridae (fairyfly), comprise an unconventional porous structure with bristles emanating from an inner core. Despite the relatively large porosity of this structural layout, the wing can generate sufficient lift to enable a rich flight dynamics. The presence of the bristles seems to be a key element in the clap and fling flight dynamics at low-to-moderate Reynolds numbers, due to a non-trivial coupling between the deformation of the bristles, and the low-to-moderate Reynolds number flow, which can make a largely porous wing behave as an impervious one. Here, we study the morphology of bristled wings to rationalize the influence of their geometrical and mechanical properties on the interactions with the surrounding fluid. Specifically, we make use of micro-fabrication techniques to manufacture deformable hairy strips, which are the simplest synthetic counterpart of bristled wings. Then, we perform a systematic exploration of the geometry of the bristle-arrangements to understand how a specific combination of porosity and permeability is optimized in insect flight. We hope that our findings can provide a better understanding of the morphology of insect wings, together with their interplay with the surrounding fluid.

1:58PM L27.00002 Bristled wings in fling: effects of varying wing kinematics , VISHWA KASOJU, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Flight capable tiny insects of body lengths <2 mm, such as thrips and fairyflies use wing-wing interaction (‘clap and fling’) to augment lift generation at chord-based Reynolds number (Re) on the orders of 1-10. In addition, these insects possess wings with closely-packed long bristles at the fringes. Previous studies have shown that bristles can reduce drag needed to fling by leaking flow through the gaps. These studies used combinations of rotational and translational motion to prescribe wing kinematics, due to lack of high-magnification video recordings of freely flying tiny insects. We experimentally investigated the effects of varying wing kinematics (pure rotation, pure translation, and overlapping rotation and translation) on force generation and leakiness during fling of a bristled wing at Re = 10. The results show that (i) drag decreased during pure rotation of the wing, (ii) peak drag increased with increasing overlap, and (iii) decreasing initial inter-wing spacing at the start of fling reduced the strength of the trailing edge vortex, resulting in asymmetric leading and trailing edge vortices.

2:11PM L27.00003 Optimal aerodynamic design of a wing with bristles , SEUNG HUN LEE, MINHYEONG LEE, DAEGYOUm KIM, KAIST — The smallest flying insects such as a fairyfly or a thrips living in a low Reynolds-number environment have evolved a bristled wing, a wing with several bristles on a thin main frame. Previous studies on the aerodynamic characteristics of a bristled wing have revealed that the gap width and the Reynolds number based on the chord length or the bristle diameter are important parameters that determine the aerodynamic performance of a bristled wing. However, these two important parameters have been treated independently thus far although they are strongly coupled aerodynamically. To examine the combined effects of the gap width and the Reynolds number, we numerically investigate a two-dimensional bristled wing with wide ranges of the gap width and the Reynolds number for a given number of bristles. With some interesting characteristics of viscosity-dominant flow, we introduce a new dimensionless parameter and propose analytic methods to estimate the aerodynamic force generation of a bristled wing for arbitrary Reynolds number and configuration.

2:24PM L27.00004 Stability of a falling bio-inspired bristled disk , MINHYEONG LEE, SEUNG HUN LEE, DAEGYOUm KIM, KAIST — Unlike most flying insects whose wings are covered by membranes, some of the smallest flying insects, Thysanoptera and Mymaridae, have wings that consist of bristles attached to a central frame. Due to their microscopic size, the smallest flying insects live in a very low-Reynolds-number regime of the order of 10 or less. Previous researches on the bristled wing have focused on finding the effects of the gaps on aerodynamic performance under various kinematics of the wing. Interestingly, tiny insects possessing the bristled wings have been reported to perform parachuting, one of passive flight modes. However, the dynamics of the bristled wing during parachuting and their effects on stability have not been studied yet. In this study, we examine the stability of a freely falling bristled disk experimentally in the low-Reynolds-number regime by changing the number of bristles and an initial falling orientation and compare with those of a full circular disk without bristles. Our experiments show that a full disk undergoes large disturbance in its orientation and displacement at the initial transient phase of free fall while a bristled disk shows more stable motion explicitly.

2:37PM L27.00005 A Novel Cylindrical Clap-and-Fling Maneuver by Swimming Marine Snails , FHERHAT KARAKAS, ALI AL DASOUQI, University of South Florida, AMY MAAS, The Bermuda Institute of Ocean Sciences, DAVID MURPHY, University of South Florida — Many insects use the Weis-Fogh clap-and-fling maneuver once per wingbeat to generate lift. Lighthill (1973) hypothesized that using this mechanism twice per stroke would create two circular vortex rings, thereby maximizing downward momentum per unit kinetic energy. We show via high speed stereophotogrammetry and micro-PIV that several pteropod species, both shelled and shell-less, do indeed use a variation of this maneuver twice per wingbeat. These pteropods flap their highly flexible wings 180° in both posterior and anterior directions so that their wingtips overlap at the end of both half-strokes to create a cylinder. The animal forces water downwards in a jet as this cylinder forms (the ‘clap’). As the wings then begin the next half-stroke, the cylinder transforms into a cone with the narrow end along the wings’ trailing edges. This cylindrical clap-and-fling maneuver induces downward flow into the cone and forms a lift-enhancing vortex ring around the wings’ leading edges (the ‘fling’). We discuss implications of performing the clap-and-fling maneuver with this cylindrical geometry versus the classic version used by insects. Further, we present preliminary results from a soft robot pteropod wing being developed to study the fluid dynamics of cylindrical clap and fling.
2:50PM L27.00006 High speed Schlieren photography on small insects , DAVID GALARZA, YUN LIU, Purdue University Northwest — The high-speed Schlieren photography has been successfully implemented in studying large flying insects like Hawkmoth Manduca, showing some promising capabilities of capturing complex three-dimensional flow structures on flying insects. To further extend our study to a wider range of insect species, like fruit fly particularly, new approach of introducing density gradient is introduced in this work for studying the flow on small insects. A preset temperature gradient is achieved inside a clear acrylic container from using a constant high temperature iron plate at the top to elevate the air temperature evenly from the top to bottom. Then an electric magnetic tethered fruit fly is carefully introduced into the container using a pole. The fruit fly is de-tethered to set for free fly afterward and flying fruit fly disturbs the evenly distributed temperature/density/gradient ambient that later can be visualized by a two-mirror parallel light high-speed Schlieren photography system.

Monday, November 25, 2019 1:45PM - 3:03PM -
Session L28 Particle Laden Flows: Deformable Particles 610 - Alfredo Soldati, TU Wien

1:45PM L28.00001 A minimal stochastic model capturing intermittency effects on coagulation in turbulence , ALAN KERSTEIN, Consultant, STEVEN KRUEGER, University of Utah — A recent theory, confirmed by direct numerical simulation, established that intermittency dominates the early transient period of zero-inertia particle coagulation in inertial-range turbulence [1]. Namely, starting from a monodisperse system, the i-mer population increases as a power of time, where the i-dependent exponent involves the third-order scalar structure-function exponent due to the connection between particle number density and an inertial-range passive-scalar field. In effect, scalar intermittency boosts the likelihood that a pairwise coalescence event is promptly followed by another due to 3-particle (hence third-order) correlations. The following minimal model reproduces the exponent values and related properties. The system state is a list of particle locations. The triplet, a measure-preserving map often used in 1D simulations and newly extended to 3D by applying it to planar slabs instead of line intervals [2], displaces particles within a given slab along the (x, y, or z) coordinate normal to the slab faces. Sampling of map occurrence times and slab orientations, locations, and widths incorporates inertial-range phenomenology. [1] J. Bec et al., Phys. Rev. E 93, 031102(R) (2016). [2] S. K. Krueger, A. R. Kerstein, J. Adv. Model. Earth Sys. 10, 1858-1881 (2018).

1:58PM L28.00002 Dynamics of chains of deformable particles in strongly confined Poiseuille and Couette flows1, SAGNIK SINGHA, Texas Tech University, ABHILASH REDDY MALIPEDDI, George Washington University, MAURICIO ZURITA-GOTOR, Universidad Loyola Andalucia, KAUSIK SARKAR, George Washington University, JERZY BLAWZDZIEWICZ, Texas Tech University — In a strong confinement system of deformable drops shear flow triggers their rearrangement into highly ordered linear arrays oriented in the flow direction. In our recent investigation [Soft Matter, 2019, 15, 4873-4889] we have found that the drop arrays behave like strongly overdamped bead-spring chains, with springs representing effective inter-drop hydrodynamic interactions. As a result, the relaxation of perturbed chains is diffusive. This behavior is in contrast to the drop-chain dynamics in a confined Poiseuille flow, which is described by the first-order wave equation. To elucidate this difference, we analyze how elementary contributions of inter-particle interactions, i.e., (i) dipolar, (ii) quadrupolar, and (iii) swapping-trajectory effects influence the collective drop dynamics. Due to antisymmetry with respect to the flow reflection, the Hele–Shaw dipoles contribute to wave propagation in a particle array. The symmetric Hele–Shaw quadrupoles together with the swapping trajectory effect produce diffusive relaxation in Couette flow and either wave decay or growth in Poiseuille flow.

1NSF Grant CBET 1603027

2:11PM L28.00003 Breakup of bubbles driven by vortex ring collision , YINGHE QI, CARL URBANIK, NOAH CORBITT, ASHWANTH SALIBINDLA, RUI NI, Department of Mechanical Engineering, Johns Hopkins University — We present an experimental investigation of bubble breakup at the moment when two vortex rings collide with each other head on at high Reynolds number, characterized by the vortex cores break into finer scales, bubbles will experience strong fluctuations of local shear and pressure at multiple length scales, reproducing a flow environment that bubbles tend to experience in fully-developed turbulence. In this study, we use the piston-cylinder arrangement to produce and control the vortex ring collision, and the timing of bubble injection is adjusted to vary the distance between bubbles and the location where two vortex cores touch each other. Four high-speed cameras are used to simultaneously measure both the bubble breakup process as well as the surrounding flow. This study will help us to explore the idea of bubble-eddy collision that has been widely used in describing bubble deformation and breakup in fully-developed turbulence.

2:24PM L28.00004 Dynamics of large and deformable bubbles in turbulence , LABANCA GABRIELE, TU Wien, GIOVANNI SOLIGO, ALESSIO ROCCON, ALFREDO SOLDATI, TU Wien; University of Udine — The dynamics of large, deformable bubbles in a turbulent channel flow is investigated coupling direct numerical simulations of turbulence with the phase-field method (PFM). In the framework of the PFM a marker function (phase field) defines the local concentration of each phase; the phase field is uniform in the bulk of the phases and undergoes a smooth transition across the interface. All fluid properties are defined as proportional to the phase field. An interfacial term (based on the Korteweg tensor) in the Navier-Stokes equation accounts for the effects of a deformable interface on the flow field. We will present the effects of density and viscosity contrasts between the dispersed and the carrier phase at a constant shear Reynolds number, Reτ = 300 (ratio between inertial and viscous terms, defined on the channel half-height), and at a constant Weber number, We = 1.5 (ratio between inertial and surface tension forces). In particular, the effects of density and viscosity contrast between the two phase on the dispersed phase morphology (drop, Sauter mean diameter, interfacial area) and the local flow field will be investigated and detailed.

2:37PM L28.00005 ABSTRACT WITHDRAWN

2:50PM L28.00006 Measurements of Deformations of Fibers and Multi-armed Particles in Fluid Flows1, BARDIA HEJAZI, LEE WALSH, GREG VOTH, Wesleyan University — We measure the deformation of fibers and particles made of 3 and 4 slender arms. The multi-armed particles are triads (3 symmetric arms in a plane) and tetrads (4 symmetric arms separated by the tetrahedral angle) with rigid arms connected near the center of the particle by a weak rubber joint. For fiber-like particles we used two arms joined at the center by a weak joint. The particles are 2cm in diameter and are 3D printed using a soft rubber-like polymer. We show that multi-armed particles deform at lower order of the aspect ratio of the arm in uniform velocity gradients compared to buckling and bending of fibers. We first examine triads in the turbulent flow of a vertical water tunnel and use 4 high-speed cameras to achieve high precision in measuring particle orientations and arm deformations with an uncertainty of roughly 10° rad. Multi-arm particles deform more readily than fibers, but the measured deformations in the water tunnel are still small and only slightly larger than our measurement uncertainty. We continue our study by measuring the deformation of fibers and tetrads in a Taylor-Couette apparatus. In these experiments we have higher velocity gradients and use a more viscous fluid than water which allows us to measure larger deformations.

1 This work was supported by NSF grant DMR-1508575.
1:58PM L29.00002 A multiphysics computational model for design and optimization of drug transport in an endovascular Chemofilter device\(^1\), NAZANIN MAANI\(^2\), TYLER C. DIORIO, Biomedical Engineering, Purdue University, West Lafayette, IN, STEPHEN W. HETTS, Radiology and Biomedical Imaging, University of California, San Francisco, CA, VITALIY L. RAYZ, Biomedical Engineering, Purdue University, West Lafayette, IN — The effectiveness of Intra-arterial chemotherapy (IAC) is limited by the majority of drugs, e.g. Doxorubicin (Dox), that pass into systemic circulation and cause cardiac toxicity. These excessive drugs can be captured by the Chemofilter (CF) — a 3D-printable, catheter based device deployed in a vein downstream of the liver during IAC. The CF chemically adsorbs Dox via ion-exchange with a surface-coated resin. In this study, the CF hemodynamic performance and drug transport were evaluated with multiphysics computational modeling. Device design was optimized using a sensitivity analysis of flow and geometry parameters. The electrochemical binding was modeled based on concentrated solution theory, where diffusion and ion migration were incorporated into an effective diffusivity term. The Navier-Stokes and Advection-Diffusion-Reaction equations were coupled in ANSYS Fluent. The optimized CF consists of an array of hexagonal channels that are twisted, perforated, and aligned with the flow direction to enhance mixing and drug binding. The optimized CF results in a 66.8% drug reduction and pressure drop of 3mmHg, with model validation by in-vivo porcine studies. These results demonstrate the utility of the CF in improving IAC performance while preventing flow stagnation regions.

\(^1\)NIH NCI R01CA194533 award
\(^2\)Presenting Author

2:24PM L29.00004 Computational modeling of bacterial dynamics under shear flow\(^1\), KARTIK JAIN, CHRISTOPH LOHRMANN, CHRISTIAN HOLM, Institute for Computational Physics, University of Stuttgart, Germany — Bacteria can propel, proliferate and accumulate in a number of media and their dynamics are affected by shear flow. In addition to self-propulsion, bacteria can stick to each other and to surfaces where they can create fast growing colonies, called biofilm. Studies have shown that bacteria more commonly accumulate at surfaces and around obstacles. In the present work we modeled dynamics of E.Coli in a water like fluid. In the model the bacteria are represented by molecular dynamics (MD) particles while the fluid is represented by a lattice Boltzmann method (LBM). The MD particles are coupled to the LB fluid using a frictional point coupling that ensures momentum conservation of the total system. We present collective transient dynamics of up to 4000 bacteria in porous configurations. Our results show that the bacteria tend to a state of momentary stasis in regions where the underlying fluid recirculates, and thus result in a sticking behavior near the obstacles of the porous media. Our findings indicate that such a behavior is manifested mainly due to hydrodynamics interactions between a bacterium and the surfaces. Our model of bacterial replication agrees well with experimental data and ongoing work includes coupling of time scales in bacterial life cycle to model biofilms.

\(^1\)Funding provided by the DFG Collaborative Research Center SFB1313
2:37PM L29.00005 Efflux Pumping Mechanism of AcrB in Multidrug Resistance Bacteria. QUYNH DINH, JIN LIU, PRASHANTA DUTTA, Washington State University — The AcrB is a multidrug pump in gram-negative bacteria responsible for ejecting antibiotics and other metabolic waste for its survival. Thus, understanding the expelling mechanisms of AcrB is essential in designing successful drugs. Though crystal structure of AcrB can be captured by the X-ray diffraction, the molecular details of AcrB-drug interaction are difficult to be identified by experiment. We have developed a hybrid coarse-grained molecular dynamics (MD) model to capture conformational changes of AcrB in presence of drug candidates at the distal binding pocket to understand the efflux mechanism. Binding energy of drugs with AcrB and potential of mean force (PMF) on drugs along the expelling pathway are also calculated. Our MD simulations show that AcrB monomer changes from binding state to extrusion state in the presence of drug or other substrate, if transmembrane residues Asp407 and/or 408 are being protonated by an antiport mechanism. Moreover, computed binding energy and PMF reveals that inhibitor-like drugs bind with AcrB much stronger than metabolic waste or conventional antibiotics. Our study suggests that inhibitor-like multifunctional drugs could effectively block AcrB pumping.

2:50PM L29.00006 ABSTRACT WITHDRAWN

Monday, November 25, 2019 1:45PM - 3:16PM — Session L30 Biological Fluid Dynamics : Cardiac Flows I 612 - Niema Pahelevan, USC

1:45PM L30.00001 Designing a Low Reynolds Number Ejector Pump for Infants with a Single Ventricle. DONGJIE JIA, MAHDI ESMAILY, MATTHEW PERONI, Sibley School of Mechanical and Aerospace Engineering, Cornell University, TIGRAN KHALAYAN, Dept of Cardiothoracic Surgery, Stanford University School of Medicine — Simulations have shown modifying stage-1 operation on single ventricles to incorporate an ejector pump presents several advantages, including improved oxygen delivery and low heart load. However, the efficient operation of an ejector pump relies on sufficient mixing to transfer energy and pump low-pressure fluid. Thus, the standard design of an ejector pump cannot operate effectively in infants where the blood flow Reynolds number is low. To address this issue, we present a novel design of an ejector pump that can effectively operate at a low Reynolds number environment. The performance of this ejector pump will then be demonstrated through implementation in the assisted bidirectional Glenn procedure that treats Hypoplastic heart syndrome. The further potential use of the device will be discussed.

1:58PM L30.00002 Artificial Right Atrium Design for Univentricular Heart Patients. HENG WEI, University of Southern California, CYNTHIA HERRINGTON, JOHN CLEVELAND, Children’s Hospital Los Angeles, VAUGHN STARNES, NIEMA PAHELEVAN, University of Southern California — Infants born with single ventricle pose a large challenge. The Fontan operation for univentricular heart patients creates a unique circulation whereby systemic veins get connected to the pulmonary arteries without passing through the cardiac chambers. As getting older, individuals with single ventricle tend to develop long-term complications like heart failure and require heart transplant. Previous studies have attempted to mechanically support these patients with standard left ventricular assist devices (LVAD). The primarily difficulty in establishing mechanical support for Fontan is that there is no blood reservoir in the closed Fontan circulation. An artificial right atrium is one of the treatments that could be implanted into the Fontan graft and provide a reservoir for blood and allow for full circulatory support. We investigated the optimum geometrical design of artificial right atrium by minimizing the particle residence time to reduce the chance of blood stagnation and clotting. Non-Newtonian Fluid-Structure Interaction (FSI) simulations employing Lattice-Boltzmann and immersed boundary method were utilized to evaluate Fontan hemodynamics. Our results indicate that the artificial atrium optimum shape is similar to the real human atrium.

2:11PM L30.00003 Is 3D Measurement Necessary for Quantifying Fluid Mechanics in a Left Ventricle? ZHENGJUN WEI, KESHAV KOHLI, YINGNAN ZHANG, VAHID SADRI, IKECHUKWU OKAFOR, AJIT YOGANATHAN, Georgia Institute of Technology — Left ventricle (LV) fluid mechanics has been broadly investigated in vitro, in vivo, and in silico over the past few decades. Previous studies have successfully demonstrated clinical relevance for hemodynamic metrics of the LV, e.g., energy dissipation, washout time, and vortex formation time, etc. A majority of these studies extracted conclusions based on 2D measurements. Additionally, 2D phase-contrast magnetic resonance imaging (PC-MRI) and echocardiography are the most common measurement tools in clinical practice. The current 3D tools do not meet the clinical demand. Unfortunately, previous literatures marginally discussed the validity of reducing the order of LV hemodynamic metrics (from 3D to 2D). In this study, an in silico LV model is developed and firstly validated against 4D PC-MRI in an in vitro, patient-specific, 3D LV phantom. Then, the settings of the in silico model are adjusted to mimic diseased states. The center-plane of the 3D phantom is extracted to represent a 2D measurement plane. The necessity of 3D measurement is elucidated based on the comparison between the LV hemodynamic metrics obtained from 3D and 2D measurements.

2:24PM L30.00004 Theoretical and Experimental Evidence for the Role of Viscosity in Ventricular Flow. ALEJANDRO ROLDN-ALZATE, RYAN PEWOWARUK, University of Wisconsin - Madison — In the ventricular flow literature, viscosity is a frequently reported parameter that is credited for being a key marker of blood flow dynamics and particularly flow efficiency. However, the exact role of viscosity in ventricular flow efficiency has yet to be explained. We apply the concept of enstrophy from turbulence and geophysical fluid dynamics to ventricular flow to explain the relationship between vorticity and viscous energy dissipation. Theoretically, enstrophy predicts a quadratic relationship between vorticity and viscous energy dissipation. Magnetic resonance velocimetry is performed in pigs using a high spatial resolution, radial acquisition (PC-VIPR). Velocity derived vorticity and viscous energy dissipation in both the left and right ventricle of pigs show strong agreement with the theoretical quadratic relationship ($R^2 = 0.94$). This work is the first rigorous explanation of the importance of vorticity in ventricular flow and we hope the concept of enstrophy is further applied in cardiovascular research. Additionally, experimental evidence shows strong agreement with theory, highlighting the ability of magnetic resonance velocimetry to quantify key aspects of cardiovascular fluid dynamics in living subjects.
2:37PM L30.00005 Cardiac Triangle Mapping: A Novel Systems Approach for Non-invasive Evaluation of Left Ventricular End Diastolic Pressure. NIEMA PAHLEVAN, MELISSA RAMOS, RAY MATTHEWS, University of Southern California — Left ventricular end diastolic pressure (LVEDP) is an important measure of global left ventricle (LV) function. Elevated LVEDP is indicative of poor LV function in both heart failure with preserved ejection fraction and in heart failure with reduced ejection fraction. This highlights LVEDP’s importance as quantitative biomarker for diagnosis, chronic monitoring, and evaluating response to therapy. Here, we introduce a new systems approach, called Cardiac Triangle Mapping (CTM), for non-invasive and instantaneous measurement of LVEDP. CTM uses arterial pressure waves and ECG to map the global ventricular function; hence, allowing computation of LVEDP. The accuracy and validity of CTM have been shown using prospective clinical data. Here, we present the validity and accuracy of CTM method using data from a prospective clinical study at the Keck Medical Center of USC.

2:50PM L30.00006 Patient-Specific Computational Fluid-Structure Interaction (FSI) Modeling of Full Cardiac Cycle1, MOHAMMAD MEHRI, MATTHEW FIG, MAYSAM MOUSAVIRAAD, University of Wyoming — A patient-specific computational fluid-structure interaction (FSI) model of full-cycle cardiac dynamics is presented. One-way and two-way coupling simulations are carried out for 2D and 3D geometries. The patient-specific 3D left ventricle (LV) geometry is constructed from 2D echocardiography images based on biplane ellipsoid model. One-way coupling studies use the wall motions and the pressure-volume (PV) loop data to specify the solid and fluid boundary conditions. The two-way coupled simulations model the myocardium passive dynamics with anisotropic Mooney-Rivlin method. Active contractions are modeled by stiffening and softening the myocardial material properties calculated as part of the one-way studies. The compressibility of blood is included in the model to produce the entire curve of the PV loop. The wall motions and PV loop data are used to validate the two-way results. Next steps will include invariant-based orthotropic models for passive behavior of myocardium and electrophysiology models for active contractions. The valvular dynamics will also need to be included for improved vorticity dynamics modeling.

3:03PM L30.00007 Elastohydrodynamics of the heart, VAMS SPANDAN, EMMANUELI VIROT, LAUREN NIU, Harvard University, WIM VAN REES, MIT, L MAHADEVAN, Harvard University — Animal hearts are deformable shells pumping large volumes of blood which guarantees oxygen for cells. Here we present a scaling for the heart rate based on a simple idea: Elastohydr

Monday, November 25, 2019 1:45PM - 3:16PM — Session L31 Biological Fluid Dynamics: Micro-swimmer Experimental

1:45PM L31.00001 Bacterial ridesharing: swimming and rolling of a sessile-motile aggregate, BIN LIU, YU ZENG, Department of Physics, University of California, Merced — While motile microorganisms disperse actively in aqueous environments by exploiting locomotory organelles, sessile cells are deficient in locomotory organelles and disperse passively through flow entrainment. Beyond these two discrete classes of motility, we explore an assembly containing both motile and sessile cells, a sessile aggregate of Caulobacter crescentus. Despite its predominantly sessile cells, the C. crescentus rosette exhibits surprisingly active motility, powered by as little as a single flagellar motor. In addition, proximity to a solid surface promotes rolling movements along the solid-liquid interface. This rolling mechanism emerges from a division of labor between sessile and motile compartments that respectively function as structural and powering modules, which can be extended to a wide range of natural and engineered microbial systems.

1:58PM L31.00002 Navigating through complex networks by sniffing gradients: diffusiophoresis vs. chemotaxis, JINZI MAC HUANG, TANVI GANDHI, ANTOINE AUBRET, DESMOND LI, SOPHIE RAMANANARIVO, MASSIMO VERGASSOLA, JEREMIE PALACCI, University of California San Diego — At the beginning of life, searching for food and evading hazards are two essential activities for microorganisms to survive, and the way they navigate is through chemotaxis. The optimal chemotaxis in complicated terrains determines the fate of living creatures, and natural selection ensures the existence of such an optimization. In our study, we investigate the navigation of inert particles in a network that has multiple junctions. In micro-networks manufactured through photolithography, a background gradient of salt is established as the signal of chemotractant by placing a source and a sink of salt. Colloidal particles then follow this signal through diffusiophoresis and move towards the source. Through stochastic modeling, we show that particles prefer to exit each junction at the end with higher concentration gradient. This preference is further enhanced when the particle size is larger, which leads to a way to magnify small signals in a network so that the colloidal particles larger than a critical size can always move towards the source of salt through the shortest path. Ultimately, we compare the navigation schemes of inert particles and living organisms, aiming to understand biological chemotaxis and shed light on future manufacturing of navigable microswimmers.

2:11PM L31.00003 Alleviation of hypoxia by biologically generated mixing from aggregations of centimeter-scale swimmers1, ISABEL HOUGHTON, University of San Francisco, JOHN DABIRI, Stanford University — Daily vertical migrations of zooplankton have been shown to affect nutrient distributions and dissolved gas concentrations in ocean and lake environments. Additionally, laboratory experiments have demonstrated the potential for mixing generated by these migrations to alter the physical structure of a water column by mixing different density water. In this work, we investigate the importance of biologically generated mixing relative to other processes in determining the biogeochemical structure of a water column inhabited by migrating zooplankton. Specifically, we consider oxygen, a highly ecologically relevant scalar, and the competition between metabolic consumption and biogenic mixing in a stably stratified water column with a hypoxic layer. We illustrate the potential for migrating animals to alleviate hypoxia, introducing complex feedbacks between the presence of animals and the biogeochemical state of their surroundings. Furthermore, we demonstrate the feasibility of oxygen as a potential indicator of biogenic mixing for future in situ investigation given its low diffusivity and higher signal-to-noise.

---

1 NSF
2:24PM L31.00004 Surfing the Wave: How Bacteria Migrate through Porous Media  
TAPOMOY BHATTACHARJEE, Andlinger Center for Energy and the Environment, Princeton University, DANIEL AMCHIN, FELIX KRATZ, SUJIT DATTAR, Department of Chemical and Biological Engineering, Princeton University — While bacterial motility is well-studied for motion on flat surfaces or in unconfined liquid media, most bacteria are found in heterogeneous porous media, such as biological gels and tissues, soils, and sediments. However, it is unknown how pore-scale confinement alters bacterial motility, because the opacity of typical 3D media precludes direct visualization. Here, we reveal that the paradigm of run-and-tumble motility is dramatically altered in a porous medium. By directly visualizing individual E. coli, we find a new form of motility in which cells are intermittently and transiently trapped as they navigate the pore space; analysis of these dynamics enables prediction of single-cell transport over large length and time scales. Moreover, we show that concentrated populations can collectively migrate through a porous medium despite being strongly confined by chemotactically surfing a self-generated nutrient gradient. This behavior depends sensitively on pore-scale confinement, initial colony density, and nutrient consumption, providing a means to control collective migration in bacterial populations. Our work provides a revised picture of active matter transport in complex media, with implications for healthcare, agriculture, and bioremediation.

2:37PM L31.00005 Controlling topological defects in Living Liquid Crystals  
NURIS FIGUEROA-MORALES, The Pennsylvania State University, ANDREY SOKOLOV, Argonne National Laboratory, MIKHAIL M. GENKIN, Cold Spring Harbor Laboratory, IGOR S. ARANSON, The Pennsylvania State University — A realization of active nematics has been conceived by combining swimming bacteria and a lyotropic liquid crystal. The complex dynamics of such active material arises from the non-trivial interplay between hydrodynamic flows and elastic forces: while bacteria are guided by the local director field, the local alignment of the liquid crystal is disturbed by the swimming bacteria. At high bacterial concentration, the domination of bacterial activity leads to creation of motile topological defects, which alter bacterial distribution. Here, we experimentally explore the possibility of controlling and pinning of emerged topological defects by artificially created microstructures. The microstructures were printed using a state-of-art multiphoton 3D lithography system and mimicked the shape of defects cores. While 1/2 defects may be easily pinned to the created pattern, -1/2 defects remain mobile. Due to an attraction between opposite defects, positive defects remain in the vicinity of pinned negative defects, significantly diminishing their diffusivity.

2:50PM L31.00006 Experimental investigation of hydrodynamic interactions between motile green algae  
JUNAID MEHMOOD, ABEL-JOHN BUCHNER, KOEN MULLER, DANIEL TAM, Delft University of Technology — The motility of micro-organisms plays a crucial role in many biological processes, such as reproduction and biofilm formation. Mechanical interactions between swimming cells can lead to collective phenomena, e.g., bio-convention or pattern formation. Hydrodynamic interactions between swimming cells have been the focus of several numerical studies, but experimental evidence of intercellular hydrodynamic interaction remains scarce. Here, we use a unique multi-camera microscopy set-up to track a dilute suspension of the model green alga Chlamydomonas reinhardtii. The cells are free to swim within a flow cell, which does not constrain their dynamics. The resulting three-dimensional trajectories provide data by which to examine pair-wise interactions between the motile cells. Hydrodynamic interactions can lead to a change in direction or velocity magnitude. This information is used to find out the length and time scale for which interaction is occurring between the pair of swimmers. We also study the velocity correlations between all the swimmers to find out the extent of length and time scale associated with these interactions.

3:03PM L31.00007 Self-organization of active spinners in liquid metamaterials  
JEAN-BAPTISTE GORCE, HUA XIA, NICOLAS FRANCOIS, HORST PUNZMANN, MICHAEL SHATS, Australian Natl Univ — The new concept of liquid metamaterials opened new ways of engineering fluids with adjustable properties at microscales. The liquid-interface metamaterials are generated by the superposition of two orthogonal standing surface waves. Fluid particles exhibit different type of trajectories depending on the phase shift between the two orthogonal waves. Circular orbits can be created when the two orthogonal waves are phase shifted by 90 degrees. The liquid metamaterials are periodic arrays of unit cells, somewhat analogous to optical lattices. The similarity with optics triggered recent investigation into the trapping of surface magnetic spinners within the liquid metamaterials. A single spinner exhibits stable orbits in a unit cell and can be guided by changing the spinning frequency and the spin direction. The spinner’s motion arises from the coupling between the spinner angular momentum and the wave angular momentum. Multiple spinners self-organize into stable configurations around the centre of the unit cells. The results offer novel methods of manipulation and confinement of actively moving surface particles at the fluid interfaces using waves and suggest new analogies between surface wave physics and confinement of nano- and micro-particles and atoms by optical fields.

Monday, November 25, 2019 1:45PM - 3:16PM –
Session L32 Biological Fluid Dynamics: Single Cells and Bacteria III 614 - Gwynn Elfring, University of British Columbia

1:45PM L32.00001 Effects of Extracellular Transforming Growth Factor-mediated Fibroblast Activation in Tumor Microenvironment  
ROBERT DILLON, ADNAN MORSHEID, PRASHANTA DUTTA, Washington State University — The transforming growth factor (TGF) is known to prevent differentiation of benign tumor cells to malignancy. Paradoxically, cancer cells exploit the immune regulation and microenvironment modulatory functions of TGF to their advantage. This makes TGF response highly sensitive to tumor progression. In vitro experimental data for several intercellular and cell surface reactions involving the fibroblasts indicate a switch-like behavior based on extracellular TGF conditions. We modeled a single tumor cell with surrounding fibroblasts to create a specific tumor microenvironment. The extracellular transport through advection, reaction and diffusion as well as cell surface reactions are captured through an immersed interface approach. Flow of extracellular nutrients and fluid-structure interactions are modeled with an immersed boundary description. The unknown reaction parameters were estimated using Bayesian inference with experimental data on the PE25 cell line. Variation in spatial distance and arrangement of tumor cells and fibroblasts showed significant change in reaction dynamics and intracellular TGF production. Microfluidic results also highlight the clinical relevance and therapeutic potential of TGF/Smad pathway.

1This work was supported by National Science Foundation under DMS-1317671
1:58PM L32.00002 Molecular Simulations of Synaptotagmin-like Protein-4-a during the Vesicle Docking and Fusion with Endothelial Cells upon Ca\(^{2+}\) Binding \(^1\), JIN LIU, QUYEN DINH, PRASHANTA DUTTA, School of Mechanical and Materials Engineering, Washington State University, Pullman WA USA — Synaptotagmin-like protein-4-a (Slp4-a) is a calcium sensor protein which plays critical roles in triggering the vesicle docking and fusion with blood-brain barrier endothelial cells during the exocytosis process. Upon binding Ca\(^{2+}\), Slp4-a undergoes a series of global translational/rotational movements and conformational changes, actively interacts with the SNARE complex, penetrates into the membrane bilayer, and triggers the pore opening. The exact molecular mechanism of how Ca\(^{2+}\) binding to Slp4-a leads to vesicle-cell fusion is not fully understood. In this work, we implement a hybrid coarse-grained force field that couples the united-atom protein models with the coarse-grained MARTINI water/lipid, to investigate the responses of Slp4-a upon Ca\(^{2+}\) binding. The hybrid coarse-grained molecular simulations enable us to explore large scale protein changes while retaining detailed molecular interactions. Our simulation results show that the binding of calcium ions causes dramatic reorientation and structural reorganization of Slp4-a. These changes induce local re-arrangement of membrane lipids at the Slp4-a-membrane contact areas leading to stronger attractive force between vesicle and endothelial cell membranes, which clearly indicate the initial docking and fusion process.

\(^1\)This work was supported by National Institute of General Medical Sciences of NIH under R01GM122081

2:11PM L32.00003 Investigation of the Key Parameters Impacting the Receptor Dependent Clathrin-mediated Endocytosis through Stochastic Modeling and Simulations \(^1\), MD MUHTASIM BILLAH, HUA DENG, PRASHANTA DUTTA, JIN LIU, School of Mechanical and Materials Engineering, Washington State University, Pullman WA USA — Receptor dependent clathrin-mediated endocytosis (CME) is one of the most important endocytic pathways taken by bioparticles, such as viruses and drug carriers, to enter the cells. During CME, the ligand-receptor interactions, assembly/disassembly of clathrin-coated pit ( CCP) and membrane deformation all act together to drive the internalization of bioparticles. Study of CME through experiments is significantly challenging because of the number of parameters impacting this complex biological process. In this work, we develop a stochastic computational model for the CME based on the Metropolis Monte Carlo simulations. After validation, we implement the model to systematically investigate effects of a wide range of biochemical and geometrical parameters on the overall internalization efficiency of particles. Specifically, results from our simulations demonstrate that the particle size and shape play critical roles during internalization. In addition, the ligand-receptor parameters, such as the receptor flexural rigidity/size and ligand-receptor reaction cutoff/energy, also significantly impact the internalization efficiency. Our model and simulations yield critical insights into CME and may provide guidelines for intra/transcellular drug delivery design.

\(^1\)This work was supported by National Institute of General Medical Sciences of NIH under R01GM122081 and NSF under CBET-1604211

2:24PM L32.00004 Filter Flows at Low Reynolds Number \(^1\), ANDERS ANDERSEN, Department of Physics and Centre for Ocean Life, Technical University of Denmark — Filter feeders that create flows through fibrous structures to capture prey particles are common among the plankton. Often the filter is external to the organism, and depending on the filter permeability and the overall motion of fluid and filter, the flow may or may not circumvent the filter. To capture small prey particles it may be advantageous to have small spacing between the filter elements, but at the same time, small filter spacing corresponds to low permeability and may result in filter circumvention and low flow rate through the filter. To explore this trade-off, we focus on low Reynolds number flow perpendicular to a thin, circular filter (permeable disk), and we determine an analytical solution for the flow through and around the filter. We compare the solution with the well-known solution for the flow past an impermeable disk, and we determine the dependence of the flow rate through the filter on its permeability and size. Finally, we discuss the possible biological implications of the results for planktonic organisms.

\(^1\)The project is supported by The Independent Research Fund Denmark (grant no. 7014-00033B), and the Centre for Ocean Life, a VKR Centre of Excellence supported by the Villum Foundation.

2:37PM L32.00005 Pinning and hydrodynamic coupling determine the motility pattern of interfacially trapped bacteria \(^1\), JIAYI DENG, MEHDI MOLAEI, NICHOLAS CHISHOLM, KATHLEEN STEBE, University of Pennsylvania — Fluid interfaces are unique environments for swimming bacteria, which display complex motility patterns influenced by concomitant capillary forces and hydrodynamic interactions. In 3-d suspension, P. aeruginosa swims along symmetric straight paths in run and reverse motions by altering the rotation direction of its flagellum to switch between pusher and puller modes. Near solid walls, the straight trajectories become circular paths due to the well understood hydrodynamic interactions. Trajectories of P. aeruginosa trapped at an oil-water interface, however, display a diverse set of trajectory types: fast and straight visitors, Brownian diffusive cells, stable curvy paths, and pirouettes. Which of these patterns occurs depends on the trapping state of the cell and its orientation with respect to the interface. An analysis of the curvy paths reveals that highly asymmetric trajectories occur with higher angular velocity and curvature for pullers and higher linear velocity and lower curvature for pushers. Our hydrodynamic analysis suggests that this motion is regulated by the re-orientation of the bacterial flagellum, which pivots normal to the interface in the puller mode and parallel to the interface in the pusher mode.

\(^1\)GomRI

2:50PM L32.00006 The colloidal hydrodynamics of intracellular transport \(^1\), ROSEANNA ZIA, AKSHAY MAHESHWARI, ALP SUNOL, EMMA GONZALEZ, DRÉW ENDY, Stanford University — Many representations of intra-cellular behavior rely on abstractions that do not account for how macromolecules are organized and move inside the cell. For many questions in biology and medicine these simpler models have been sufficient. However, fundamental gaps in understanding of many cell functions persist; physics may provide a bridge to close such gaps. I will discuss our computational and theoretical models of spherically confined colloidal suspensions, as a simple model cell, where biomolecules and their interactions can be physically represented individually and explicitly. A primary challenge in models of confined colloidal suspensions is the accurate and efficient representation of many-body hydrodynamic interactions, Brownian motion, and the enclosure. To this end, we developed the Cellular Stokesian dynamics framework. Utilizing this model, we studied diffusion, cooperative motion, and self-organization with confinement and crowding levels representative of a cell interior. I will discuss the qualitative influence of hydrodynamics, confinement and crowding on transport behavior, as well as the consequences of neglecting such influences. Connections to underlying structure are made, and implications for cellular function are discussed.

\(^1\)GomRI
3:03PM L32.00007 Fluid forces and flows in muscle’s contractile lattice\textsuperscript{1} SAGE MALINGEN, University of Washington, KAITLYN HOOD, ANETTE HOSOI, Massachusetts Institute of Technology, THOMAS DANIEL, University of Washington — Muscle cells are specialized for large, rapid shape change. Their contraction is powered by the collective action of molecular motors anchored to long protein filaments (thick filaments) that form a densely packed, fluid filled lattice. Molecular motors bind to thin filaments and pull them past the thick filaments. These motors are estimated to produce 5 pN forces. As filaments slide they shear with the surrounding fluid. Additionally, the lattice volume can change during contraction, causing flows that create axial and radial shear forces. With an inter-filament gap distance of only 30 nm surface-to-surface, viscous drag forces could be non-trivial and have not been measured. Using a finite element model (COMSOL) of the contractile machinery we estimate that viscous drag forces on single filaments are on the order of 10 fN. We estimate that the energy dissipated by viscous drag over all filaments is less that 1% of the energy used by the system. However additional proteins occupying the gaps between filaments, variable inter-filament spacing and viscosity values greater than that of water may influence this result.

\textsuperscript{1}Bioengineering Cardiac Training Grant (NIH/NIBIB T32EB1650), Army Research Office, Komen endowed chair & ARCS

Monday, November 25, 2019 1:45PM - 3:16PM — Session L33 Flow Instability: Geophysical

1:45PM L33.00001 The Weakly Nonlinear Evolution of Superharmonics Generated by Model Oceanic Internal Tides\textsuperscript{1} BRUCE SUTHERLAND, University of Alberta, LOIS BAKER, Imperial College London — It is now well-established that a vertically confined, horizontally periodic internal mode in non-uniformly stratified fluid self-interacts through the advection terms resulting in the forcing of superharmonics with double the horizontal wavenumber and frequency of the parent mode. The work presented here examines the linear and weakly nonlinear response to this forcing resulting from mode-1 “parent” waves in stratification typical of the ocean. It is shown that the forcing results in the excitation of a near-pure mode-1 (“sibling”) internal wave with double the horizontal wavenumber of the parent, though the natural frequency of this mode differs slightly from double the frequency of the parent. As a result, the sibling wave first grows and decays, doing so periodically at the beat frequency set by the difference of the natural frequency of the mode and double the parent-mode frequency. A weakly nonlinear analysis is necessary to predict the maximum amplitude of the superharmonic. This considers how the interaction between the parent-mode and its superharmonic sibling put energy into or take energy out of the parent-mode.

\textsuperscript{1}This research was made possible due to the National Science Foundation (Grant OCE-1332750) for their support of the WHOI Geophysical Fluid Dynamics Summer Program. Additional funds were provided to Sutherland by the Natural Sciences and Engineering Research Council (NSERC) of Canada through its Discovery Grant program.

1:58PM L33.00002 Dynamic stability of a jet near a transition in static stability JOHN MCHUGH, The University of New Hampshire — The vertical profile of the Earth’s atmosphere contains a sharp transition region at the tropopause between two roughly constant stability layers, and also jet streams at nearly the same altitude, with the jet stream core possible above or below the tropopause, depending on location. This proximity of the jet to the tropopause would be expected to greatly affect the dynamic stability of the jet, treated here with the jet modeled with the Bickley \textit{sech}\textsuperscript{2} profile and the tropopause modeled as a smooth transition region with a tanh profile. Stability results are obtained numerically using a Chebyshev collocation spectral method. The results show that the jet becomes more unstable as it is moved further beneath the tropopause. Corresponding two-dimensional direct numerical simulations of the flow confirm the initial growth rate, but then show that the most unstable mode achieves more kinetic energy when the jet is just above the tropopause. Overall, the results indicate that when a jet is above the tropopause, the configuration is more stable and more likely to produce a strong single unstable mode. Conversely, when a jet is below the tropopause, the jet is more likely to form a broad spectrum of motion.

2:11PM L33.00003 Four wave interactions for internal waves JEAN-MARC CHOMAZ, LadHyX, CNRS-Ecole polytechnique, SABINE ORTIZ, Unit de Mcanique, Ecole Nationale Suprieure de Techniques Avances, Paris, LADHYX, CNRS-ECOLE POLYTECHNIQUE TEAM, UNIT DE MCAINIQUE, ECOLE NATIONALE SUPRIEURE DE TECHNIQUES AVANCES, PARIS TEAM — Triadic instability is a very generic mechanism by which a primary wave of finite amplitude is destabilized by two secondary waves (daughter waves) forming a resonant triad. For gravity waves in the ocean, as shown by Philips, O.M. (UPC, 1967), resonant triads form several continuous branches, which can be represented in two dimensions as resonant lines in the plane of the wave vector of one of the secondary waves. We show here that the crossing of two of these branches radically modifies the nature of triadic instability by coupling, no longer two daughter waves, but three that form two triads sharing one same wave. Instability is then reduced for the triad unstable in classical theory while the second triad, stable according to classical theory, is strongly destabilized. Building on McEwan, A.D. & Plumb, R.A. (Dyn. Atm. & Oceans, 1977), we show that this modification of triadic instability affects a finite region around the crossing point of resonant branches in the plane of wave vectors, region whose extent increases very rapidly as the amplitude of the primary wave increases. The direct calculation of instability modes by a Floquet method shows that, even for a very small amplitude of the primary wave (Froude number of about 0.01), the deviation from the classical theory.

2:24PM L33.00004 Linear instability of the Prandtl model for stratified slope flows\textsuperscript{1} INANC SENOCAK, CHENG-MIAN XIAO, University of Pittsburgh — Ludwig Prandtl dedicated the last three pages in the authorized translation of his famous book “Essentials of Fluid Dynamics” to describe mountain and valley winds in stratified air. The exact solution assumes the presence of a constant linear background stratification above a sloped surface of infinite extent in all directions with a constant heating or cooling applied to it. Over the years, the Prandtl model has been found to describe the flow profile of kataibatic winds observed in nature. Despite the usefulness of the Prandtl model, its stability characteristics have gone unexplored for many decades since its publication. In the present work, we use linear modal analysis and direct numerical simulations to uncover two types of flow instabilities in the Prandtl model for anabatic and kataibatic slope flows. These instabilities manifest themselves as a function of the slope angle, the Prandtl number and a newly introduced stratification perturbation parameter, which is a measure of the relative importance of the surface heat flux with respect to the background stratification. We investigate the characteristics of these instabilities as a function of the aforementioned dimensionless parameters and highlight their differences under kataibatic and anabatic scenarios.

\textsuperscript{1}Sponsored by the Army Research Office under grant no. W911NF-17-1-0564
2:50PM L33.00006 Shear instabilities in laboratory arrested salt-wedge flows

ADAM JIANKANG YANG, EDMUND TEDFORD, JASON OLSTHOORN, GREGORY LAWRENCE, University of British Columbia — The spatial variation in the properties of an arrested salt wedge and its resulting Holmboe instabilities have been investigated, both analytically and in the laboratory. In the laboratory particle image velocimetry and laser induced fluorescence are used to obtain flow velocities and the height of the density interface. The positive and negative Holmboe wave modes are separated by the 2D Fourier transform. An analytical solution for the profile of interface height, in the absence of interfacial flow instabilities, has been developed from two-layer internal hydraulic theory. The evolution of the velocity profile is predicted using a momentum diffusion equation following a Lagrangian frame of reference along the interface of the salt wedge. The centre of the shear layer is predicted to lie above the density interface, with this offset decreasing in the downstream direction. Due to the offset and lower boundary, the growth rate of the negative instability is smaller than that of the positive instability. Our theoretical predictions are in good agreement with the laboratory measurements.

3:03PM L33.00007 Halite precipitation from double-diffusive salt fingers in the Dead Sea: Numerical simulations

RAPHAEL QUILLON, University of California, Santa Barbara, NADAV LENSKY, VLADIMIR LYAKHOVSKY, ALI ARNON, Geological Survey of Israel, ECKART MEIBURG, University of California, Santa Barbara — Thick, extensive salt layers are commonly found in the Earths geological record and formed as a result of a negative water balance in hypersaline lakes saturated in salt. Today, the Dead Sea is considered to be the only modern analog to these deep hypersaline lakes. Recent field work conducted by the Geological Survey of Israel showed that during the dry summer season, the top layer of the Dead Sea is warmer, saltier and undersaturated in salt, and that double-diffusive convection is responsible for delivering dissolved salt from the top layer to the bottom layer, resulting in continuously supersaturated bottom layer and seasonally undersaturated top layer. We present numerical simulations of this double-diffusive process directly based on measurements from the field work. The simulations account for the phase change from dissolved to crystalline salt, and for the settling of the salt crystals. We show that no other physical mechanism than double-diffusion is required in order to generate sufficient transport of salt and obtain the salinity and temperature profiles measured in the summer in the Dead Sea. The combined field measurements and numerical simulations paint a novel and promising picture for the mechanisms of salt deposition in the historical record.

Monday, November 25, 2019 1:45PM - 3:16PM —
Session L34 Flow Instability: Rayleigh-Taylor II

1:45PM L34.00001 Effect of Magnetic Fields on the Nonlinear Rayleigh-Taylor Instability

XIN BIAN, RICCARDO BETTI, HUSSEIN ALUIE, University of Rochester — The magneto-Rayleigh-Taylor instability (mRTI) plays an important role in inertial fusion schemes, including ICF and magLIF. It is also hypothesized that mRTI is pervasive in the interstellar medium, where it tends to concentrate the plasma into discrete clouds. We investigate numerically the effects of external magnetic fields on RTI during its nonlinear stages in 2D and 3D. We consider magnetic fields oriented in both parallel and perpendicular directions relative to the initial interface. Both magnetic orientations tend to suppress bubble development in 2D but can enhance it in 3D, where we find a non-monotonic dependence on field strength. For example, we observe that a perpendicular magnetic field stronger than a threshold enhances the RTI growth by enhancing the anisotropy of mixing and suppressing horizontal motions, especially at the smallest scales. Moreover, magnetic fields in either direction tend to accentuate the asymmetry between bubbles and spikes

1Supported by DOE FES grant number DE-SC0014318

1:58PM L34.00002 Direct Numerical Simulations of Magnetic Rayleigh-Taylor Instability in ICF Coasting Stage

ZHAORUI LI, Texas A&M University, DANIEL LIVESCU, Los Alamos National Laboratory — The development of hydrodynamic instabilities is generally believed to be one of the main obstacles to achieving economically controlled inertial confinement fusion (ICF). In this study, accurate simulations of 2D magnetic Rayleigh-Taylor instability (RTI) under ICF coasting stage conditions have been conducted with a newly developed high-order two-fluid plasma solver (Li and Livescu, 2019), in which full transport terms, including temperature and magnetic field dependent ion and electron heat fluxes and viscous stresses, are implemented. The numerical results show that, for the initial configuration with small Atwood number (0.33) and high hot-spot temperature (10 keV), the extremely large heat conduction and viscous stresses completely suppress the RTI development. Instead, the late-time RTI and magnetic field can grow to structures close to those seen in previous studies, which relied on numerical transport to regularize the equations, only when unrealistically small transport coefficients are used for calculating heat conductivity and viscosity. Further simulations with realistic transport phenomena are being conducted with more relevant ICF coasting stage conditions (e.g. Weber et al. 2014) in which Atwood number is 0.875 and hot-spot temperature is 2.25 keV.
The presented work focuses on the preliminary implementation of Planar Laser Induced Fluorescence (PLIF) to study the Blast-Driven Instability (BDI) in cylindrical geometry at the Georgia Tech Shock Tube and Advanced Mixing Laboratory. By using detonators to generate blast waves, a gaseous interface is subject to the combined Richtmyer-Meshkov (RMI) and Rayleigh-Taylor Instabilities (RTI); the two instabilities comprising the BDI. Previous validation of the facility was completed using high speed Mie Scattering and demonstrated faithful reproduction of the phenomena of the BDI. Previously completed Mie Scattering measurements will be used in conjunction with PLIF to measure the degree of interfacial stretching for bubble-spike pairs in this facility. These efforts will be put toward the implementation of simultaneous PLIF and PIV for full flow field measurements.

1 Authors acknowledge support from DOE/NNSA Grant DE-NA0003195 and the NSF-CAREER Award 1453056.
We observe that for k study the dynamics of arbitrary-shaped particles in a second-order fluid, subject to a general quadratic flow field. For the two model constants \( v_1 \) and \( v_2 \) (first and second normal stress coefficients) we assume the relationship \( v_1=2v_2/3 \). The assumption allows us apply a multipole expansion to derive analytical expressions for the polymeric force and torque of the particle. We apply the analytical solutions to track the translational and rotational trajectories of spherical, spheroids, and general ellipsoids in shear and pressure driven flows. In shear flows, we observe that prolate-like particles undergo a transition from tumbling to log-rolling motion (i.e., alignment along the vorticity direction) as the shear rate increases. At very large shear rates, the particles can reorient along the flow direction, but this state is metastable. In pressure driven flows, we find that particles migrate to the center of the flow, with the tumbling period increasing in time until the particle eventually aligns along the flow direction.

We vary the drop-to-medium viscosity ratio (k) by varying the PVP concentration and demonstrate center-ward droplet migration for k \( \leq 3.72 \) respectively, while droplets stay near the wall for 3.72 \( \leq k \leq 18.52 \), contrary to Chan and Leal’s prediction with second-order fluids. We observe that for k = 12, Newtonian castor oil (viscosity 650 mPa-s, relaxation time = 0) droplets migrate to center while PDMS (average viscosity 666 mPa-s, relaxation time = 0.001) droplets stay near the wall. Finally, we demonstrate viscoelasticity based sorting of castor and PDMS droplets.

1.45PM L35.00001 Cross-stream migration of non-spherical particles in a second-order fluid. SHIYAN WANG, CHENG-WEI TAI, VIVEK NARSMIHAN, Purdue University — Particle migration in viscoelastic suspensions is vital in many applications in the biomedical community and the chemical/oil industries. Despite previous studies on the motion of spherical particles in simple viscoelastic linear flows, the combined effect of more complex flow profiles and particle shapes is underexplored. Here, we study the dynamics of arbitrary-shaped particles in a second-order fluid, subject to a general quadratic flow field. For the two model constants \( v_1 \) and \( v_2 \) (first and second normal stress coefficients) we assume the relationship \( v_1=2v_2/3 \). The assumption allows us apply a multipole expansion to derive analytical expressions for the polymeric force and torque of the particle. We apply the analytical solutions to track the translational and rotational trajectories of spherical, spheroids, and general ellipsoids in shear and pressure driven flows. In shear flows, we observe that prolate-like particles undergo a transition from tumbling to log-rolling motion (i.e., alignment along the vorticity direction) as the shear rate increases. At very large shear rates, the particles can reorient along the flow direction, but this state is metastable. In pressure driven flows, we find that particles migrate to the center of the flow, with the tumbling period increasing in time until the particle eventually aligns along the flow direction.

1.58PM L35.00002 Viscoelasticity based droplet migration and sorting. SHAHMIK HAZRA, Indian Institute of Technology, Madras, SUSHANTAA MITRA, Waterloos Institute for Nanotechnology, University of Waterloo, ASHISH KUMAR SEN, Indian Institute of Technology, Madras — We experimentally elucidate the cross-stream migration behavior of viscoelastic Polydimethylsiloxane (PDMS) droplets in the non-inertial Poiseuille flow of constant viscosity aqueous viscoelastic solution of Polyvinylpyrrolidone (PVP) in straight rectangular microchannels. We investigate the complex interaction among deformability induced non-inertial lift force, viscoelastic lift force due to matrix viscoelasticity, and viscoelastic lift force due to droplet viscoelasticity and propose a new droplet migration regime. We vary the drop-to-medium viscosity ratio (k) by varying the PVP concentration and demonstrate center-ward droplet migration for k \( \leq 3.72 \) respectively, while droplets stay near the wall for 3.72 \( \leq k \leq 18.52 \), contrary to Chan and Leal’s prediction with second-order fluids. We observe that for k = 12, Newtonian castor oil (viscosity 650 mPa-s, relaxation time = 0) droplets migrate to center while PDMS (average viscosity 666 mPa-s, relaxation time = 0.001) droplets stay near the wall. Finally, we demonstrate viscoelasticity based sorting of castor and PDMS droplets.

2:11PM L35.00003 Oscillations of a cantilevered micro beam driven by a viscoelastic flow instability1, ANITA DEY, YAYHA MODARRES-SADEGHI, University of Massachusetts Amherst, ANKE LINDNER, ESPCI, CNRS, University Paris Diderot, JONATHAN ROTHSTEIN, University of Massachusetts Amherst — We will present the results of our study of the flow of a viscoelastic polymer solution past a cantilevered beam in a confined geometry. The flow of viscoelastic fluids, unlike Newtonian fluids, become unstable even at infinitesimal Reynolds numbers due to purely elastic flow instabilities that occur at large Weissenberg numbers. With increasing Weissenberg number, we will show that elastic instabilities occur in the vicinity of a flexible beam and begin to interact with the beam. We will report these interactions for cantilevered beams with varying elastic modulus, beam length and rigidity. Over a range of Weissenberg numbers, we will show that the flow field transitions from a stable detached vortex upstream of the beam to a time-dependent unstable vortex shedding. The shedding of the unstable vortex upstream of the beam will be shown to couple with the flow-induced beam deformation triggering oscillations of the beam. The critical onset of the flow transitions, mechanism of vortex shedding, and dynamics of the cantilevered beam oscillations will be presented for beams with varying flexibility. The oscillations of the flexible beam will be shown to have distinct regimes: a clear single vortex shedding regime and another regime characterized by a 3D chaotic flow instability.

1 NSF Grant CBET-1705251, ERC Consolidator Grant No. 682367 and the Institut Pierre-Gilles de Gennes, program ANR-10-EQPX-34.

2:24PM L35.00004 Two ball interaction while settling in an Oldroyd-B fluid: one atop the other initially, TSORNG-WHAY PAN, Dept. of Mathematics, University of Houston, SHANG-HUAN CHIU, Dept. of Scientific Computing, Florida State University, ROLAND GLOWINSKI, Dept. of Mathematics, University of Houston — In this talk we present a numerical study of two ball interaction while settling in a vertical channel with a square cross-section filled with an Oldroyd-B fluid. Two balls are released one atop the other initially and the effects of particle inertia and fluid inertia are not ignored. We have obtained that either the trailing ball catches up the leading one to form a chain or two balls separate with a stable final distance at the end. For the cases of the ball density slightly heavier than that of the fluid, they can form a vertical chain or tilted chain. But when increasing the ball density, the two balls can form a chain for smaller initial gaps; but they move away from each other and the distance between two balls reaches a constant for a larger initial gap. When the density of the balls is up to, at least, 3.3 times of the fluid density, the two ball settling velocity (i.e., the ball density), initial gap between two balls and elasticity number. Also the fluid polymer extension limit has its effect on the formation of two ball chain when they settle in a FENE type of viscoelastic fluid.

2:37PM L35.00005 Filament thinning dynamics of Boger fluids in extensional flow under microfluidic environment. TANOY KAHALI, SUMAN CHAKRABORTY, Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, Kharagpur - 721302, India — Polymer addition to Newtonian-fluid drastically alters the neck thinning dynamics especially in reduced length scale system due to enhanced elasticity. This motivates to study the neck thinning of two-phase polymeric–Newtonian system in micro-scale. Previous studies are mostly focused on gravity assisted filament thinning in macro-scale. Here, we intend to study the effect of elasticity on filament thickness for a series of Boger fluids(dispersed phase)stretched in a medium of silicon oil under micro-environment. Qualitatively, we observed that the filament thinning rate is much slower compared to Newtonian fluid. At an earlier stage, both fluids undergo a shearing thinning process where the filament thickness decays exponentially with time. In later stage, for Newtonian fluid a capillary driven regime dictates further thinning and rupture of the filament. In contrast, a second exponential thinning regime (causing the delay) is observed for polymer filaments along with capillary-driven thinning before pinch off. We envisage that this analysis may elucidate the role of different types of polymer addition and its concentration on the universal trend of filament thinning process in micro-scale.

2:50PM L35.00006 Viscoelastic fluid-structure interactions in microfluidics1, SIMON HANGARD, CAMERON HOPKINS, AMY SHEN, Okinawa Institute of Science and Technology — Flow of a viscoelastic wormlike micellar solution around a slender, but rigid, microfluidic post at negligible Reynolds number (Re<1) undergoes a supercritical bifurcation to a steady asymmetric state when a critical Weissenberg number (Wi) is exceeded. A second transition above a higher critical Wi results in time-dependence of the asymmetric flow with a characteristic frequency \( 1/\lambda_M \), where \( \lambda_M \) is the Maxwellian relaxation time. We examine the effect of this time dependence on the behavior of flexible cantilevered micro-posts, showing post oscillations at the same characteristic frequency \( 1/\lambda_M \), thus demonstrating a “purely-elastic” fluid-structure interaction. A second flexible post positioned downstream, shows a remarkable degree of synchronization with the first. The time lag between their correlated motions is much shorter than any expected flow time between the two posts. Our experiments show that the posts are effectively linked by an elastic strand of highly stressed fluid originating in the wake of the upstream post. Our results indicate that the time lag between their motion is dictated by the speed of the elastic wave traveling along this strand.

1 JSPS grant nos. 17K06173, 17K00412, 18K03958, 18H01135
3:03PM L35.00007 Motion of nanoparticle-covered droplet in a square microchannel

ZHENGYUAN LUO, BOFENG BAI, State Key Laboratory of Multiphase Flow in Power Engineering, Xin Jiaotong University — The flow of complex droplets with contaminated surfaces (e.g., nanoparticle-covered interface) in microchannels with non-circular cross-sections is ubiquitous in nature and various engineering applications. For example, deformable droplets through porous media in underground oil reservoir and droplet transport and manipulation in microfluidic devices. It is also an important fundamental question in the discipline of fluid mechanics. Extensive studies have been dedicated to the study of the motion, deformation and breakup of an individual droplet in cylindrical capillaries or non-circular channels, most of which have been focused on clean droplets. However, little is known about the effects of contaminated surfaces, e.g., nanoparticle-covered interface. In this study, we will show our new numerical results on flow dynamics of complex droplets with nanoparticle-covered interface in a square microchannel. We find the adsorption of nanoparticles tends to assemble at the drop rear and immobilize the drop surface, and thus enlarges the droplet-induced extra pressure loss.

1This work was supported by the National Science Fund for Distinguished Young Scholars of China (Grant No.51425603) and the Young Scientists Fund of the National Natural Science Foundation of China (Grant No.51606146).

Monday, November 25, 2019 1:45PM - 3:16PM –
Session L36 Microscale Flows: Emulsions

1:45PM L36.00001 Effect of adding dispersant on the structures of water-in-oil emulsions.

DIEGO F. MURIEL, JOSEPH KATZ, Johns Hopkins University — It is believed that dispersant application to break up marine oil slicks is effective only before stable water-in-oil emulsions develop. Both breaking waves and gentle mixing produce stable emulsions, and surfactants present in crude oil stabilize entrained sea water even in calm seas. Once formed, the interaction of these emulsions with chemical dispersant is poorly understood, partially because field observation at microscopic scales is challenging. This study examines the effect of adding dispersant (Corexit 9500) on the structure of emulsions with or without external mechanical energy. Microscopic imaging examines the size, spatial distribution, and time evolution of water droplets in the emulsion prior to and after introducing dispersant. Initially, these droplets form a multi-scale lattice with small droplets aggregating around larger ones. Adding dispersant without mixing generates secondary flows as the water droplets coalesce. In time, part of the water separates, a fraction forms a cloud of submicron droplets, and the rest remains unchanged. Agitating the dispersant-emulsion mixture enhances the phase separation, removing about 66% of the entrained water, and leaving an emulsion with finer droplets with different rheological properties (higher viscosity).

1MPRI. Multi Partnership Research Initiative, Canada.

1:58PM L36.00002 Droplet coalescence using a microfluidic hydrodynamic trap.

SHWETA NARAYAN, University of Minnesota - Twin Cities, DAVIS MORAVEC, BRAD HAUSER, ANDREW DALLAS, Donaldson Company, Inc., CARI DUTCHER, University of Minnesota - Twin Cities — Coalescence of micrometer-sized droplets to form larger drops is a fundamental process leading to separation of complex emulsions. Single droplet coalescence experiments are challenging, particularly with micrometer-sized droplets, compared to bulk studies. We have shown using microfluidics that the dynamic interfacial tension equilibration timescale is orders of magnitude shorter in micrometer-scale droplets compared to large millimeter-sized drops. Here we employ a microfluidic hydrodynamic trap to trap and coalesce single micrometer-sized droplets formed on-chip using feedback pressure control. Similar to the macro-scale four-roll mill, single droplet coalescence experiments are conducted using a microfluidic trap, with precise control over droplet size and speed. The systems studied are light and heavy mineral oils with varying concentrations of SPAN 80 in the continuous phase, with water as the dispersed phase. Film drainage times are measured as a function of Capillary number, surfactant concentration and viscosity ratio.

1This work was funded by Donaldson Company and performed at the University of Minnesota. Portions of this work were conducted in the Minnesota Nano Center, which is supported by the National Science Foundation through the National Nano Coordinated Infrastructure Network (NCCI) under Award Number ECCS-1542202.

2:11PM L36.00003 Study on the lateral migration of a ferrofluid droplet in a plane Poiseuille flow under uniform magnetic fields.

MD. RIFAT HASSAN, CHENG WANG, Department of Mechanical and Aerospace Engineering, Missouri University of Science and Technology — Droplet dispersion in another immiscible fluid is important in a variety of technological processes that involve liquid-liquid extraction where phase separation is crucial to the purification of the product. In this study, we investigate the lateral migration of a ferrofluid droplet in a plane Poiseuille flow under uniform magnetic fields by means of a numerical simulation, which uses a level set method to track the droplet interface between the two phases. Focusing on low droplet Reynolds number (i.e., $Re_d \leq 0.05$), the results indicate that the magnetic field plays a pivotal role in the motion trajectory of the droplet and the final equilibrium position in the channel. When the magnetic field acts in a direction parallel to the flow direction (i.e., $\alpha = 0^\circ$), the droplet settles closer to the bottom wall with increasing magnetic field strength, while at $\alpha = 45^\circ$ the droplet settles closer to the center. Also, variation of initial droplet sizes results in different equilibrium positions along the channel. Furthermore, at $\alpha = 90^\circ$ the droplet finds its equilibrium position at the channel center irrespective of different magnetic field strengths and droplet sizes.

2:24PM L36.00004 Bubble Nucleation and Streaming in Degasosed Water.

KYOKO NAMURA, SHUNSUKE OKAI, SAMIR KUMAR, MOTOFUMI SUZUKI, Kyoto University, MICRO PROCESS ENGINEERING LABORATORY TEAM — Marangoni flow around a microbubble has been extensively studied to apply it for microfluidic control. However, precise control of the Marangoni flow is not straightforward because of the difficulty in controlling temperature distribution around the tiny bubble. Especially, the bubble growth under heating affects the temperature gradient on the bubble surface, which is generally induced by the diffusion of the dissolved air gases into the bubble. Here we study the bubble nucleation and subsequent flow generation in well-degassed water. The ultrapure water was sonicated under vacuum (5 kPa) to remove dissolved gases. In order to realize the localized heating of the prepared degassed water, we used the photothermal effect of the gold nanosand film. By irradiating CW laser on the film immersed in the degassed water, a water vapor bubble was generated. The bubble oscillated at sub-MHz, where its maximum size was 10 μm. The bubble involved significantly rapid flow of the order of 1 m/s under continuous photothermal heating, which is expected to be useful for microfluidic mixing and pumping. The rapid flow generation is attributed to the Marangoni force and bubble oscillation, which are induced under the steep temperature gradient.

1This project has received funding from JSPS KAKENHI Grant Nos 17H04904 and 19K21932.
Using machine learning to discover shape descriptors for predicting emulsion stability in a microfluidic channel

Evaporation-triggered ouzo effect in a Hele-Shaw cell

Verification and validation of a continuous adjoint formulation for liquid-gas flows

Tracking droplet breakup in homogeneous isotropic turbulence

Interfacial Behavior of Surfactant Covered Double Emulsion in Extensional Flow

Evaporation-triggered ouzo effect in a Hele-Shaw cell

Verification and validation of a Continuous Adjoint Formulation for Liquid-Gas Flows

Tracking droplet breakup in homogenous isotropic turbulence
2:11PM L37.00003 Physics of liquid break-up in a two-fluid coaxial atomizer forced by an external acoustic field. PETER DEARBORN HUCK, RODRIGO OSUNA-OROZCO, NATHANIEL MACHICOANE, ALBERTO ALISEDA, University of Washington — We investigate acoustic forcing as a means of actuation in the formation of liquid drops and ligaments at the interface between a low momentum liquid cylinder and a high momentum coaxial gas jet. A canonical two-fluid co-axial atomizer is experimentally investigated, at a wide range of gas to liquid momentum ratios, but within the high Weber number break-up limit. The liquid cylinder placed at the velocity of an acoustic waveguide where the second traverse mode of the cavity is excited. Anemometry measurements show disturbances in the boundary layer at the nozzle orifice, but the self-similar nature of the gas-phase jet is retained in the mid-field. Phase Doppler Interferometry measurements of the droplet size, velocity and number density confirm that acoustic forcing in the near field is effective at enhancing atomization (the Sauter Mean Diameter decreases by 25%).

2:24PM L37.00004 Swirl Optimization to Maximize Spray Angle from a Coaxial Airblast Atomizer. SCOTT A. ZMUDA, TIMOTHY B. MORGAN, JULIE K. BOTHELL, THEODORE J. HEINDELM, Iowa State University, EXPERIMENTAL MULTIPHASE FLOW LAB TEAM — Liquid sprays play a key role in many engineering processes (e.g., food processing, coating and painting, 3D printing, fire suppression, combustion systems, etc.). One method of forming a spray is through a coaxial airblast atomizer in which a liquid stream is surrounded by a gas jet. Adding swirl to the gas jet can modify the resulting spray angle. This study completes a parametric study over a range of gas and liquid flow rates, momentum ratios, and swirl percentages, to identify those conditions that maximize spray angle. The spray angle is determined by back-illuminating the spray with an LED light panel to form a shadow of the resulting spray. A high-speed camera is then used to capture a series of successive images. Image analysis is then used to determine an average spray angle. It is shown that the swirl percentage for the maximum spray angle is a function of the flow conditions.

2:37PM L37.00005 Particle-droplet interaction in turbulent channel flow. ARASH HAJSHARIFI, CRISTIAN MARCHIOLI, University of Udine, ALFREDO SOLDATI, TU Wien — We examine the interaction between small (sub-Kolmogorov) inertial particles and large deformable droplets in turbulent channel flow. To simulate such solid-liquid-liquid flow, we exploit a Eulerian-Lagrangian methodology based on direct numerical simulation of turbulence, coupled with a Cauchy-Hillard Phase Field Model to capture the interface dynamics and Lagrangian tracking to compute particle trajectories. We model the particle-interface interaction via a capillary force based on the liquid-liquid surface tension and on the local interface curvature. This force gives a potential well that drives particle accumulation on the droplet interface, which acts as particle adsorber. We quantify particle-interface interaction in a simplified situation where the droplets have the same density and viscosity of the carrier liquid (mimicking a water-oil emulsion), and particles are one-way coupled with the fluids. Particle-induced deformation of the interface and the effect of gravity are neglected. Preliminary simulations indicate that particles tend to aggregate in regions of the interface characterized by high curvature, and that this tendency is modulated by particle inertia. The higher the inertia of the particles, the stronger the tendency to escape from the interface.

2:50PM L37.00006 Multi-Scale Investigation of Spray Droplets Issued in a Turbulent Background. DOUGLAS CARTER, ROUMAISSA HASSAINI, FILIPPO COLETTI, University of Minnesota — We present time-resolved planar measurements of an upward-facing hollow cone spray with simultaneous large and small scale fields of view. The measured mean droplet diameter of 50um for the spray issued into both quiescent and turbulent backgrounds is found to correspond to Stokes numbers approaching order one, indicating that the droplets act as non-ideal flow tracers. The dynamics of the drops are investigated by particle tracking velocimetry (PTV) to deduce the instantaneous droplet motion and particle image velocimetry (PIV) to extract the correlated droplet motion in the far field of the spray. It is found that the droplets issued into quiescent air exhibit greater random uncorrelated motion (RUM) compared to the droplets in a turbulent background, implying that the turbulent eddies prevent the droplets from moving ballistically. This is also reflected by the lesser drop in the Eulerian autocorrelations at small separations for droplets issued into a turbulent background as well as the general increase in the magnitude of the Lagrangian autocorrelations. These results provide novel insight into the motion of inertial droplets in a turbulent gas, and have implications for the numerical modeling and control of liquid sprays.

3:03PM L37.00007 Experimental Analysis of Spray Atomization for Targeted Deposition of Nanoparticles. SHADI SHARIATNIA, Texas A&M University, FARZAD POURSADEGH, Georgia Institute of Technology, AMIR ASADI, DORRIN JARRAHBASHI, Texas A&M University, A&M-GT COLLABORATION, A&M-GT COLLABORATION — Targeted delivery of nanoparticles has a wide range of applications in electronics, pharmaceuticals, and advanced manufacturing. A novel process that deposits cellulose nanocrystals (CNC) on a silicon wafer has been introduced. CNC-containing droplets were generated by injecting aqueous suspension of CNC through an air-atomized nozzle, where a high-pressure liquid jet undergoes severe instabilities due to the large pressure drop between the injector and ambient atmospheric conditions. High-speed diffuse back illumination (DBI), laser diffraction techniques and scanning electron microscopy (SEM) are employed to visualize and quantify the effects of spray parameters on nanoparticles distribution. The liquid jet breaks up into ligaments that later form large number of micron-size droplets at a small distance downstream of the nozzle that further evaporate and release nanoparticles on the targeted surface. Controlling process parameters such as pressure and temperature of the jet, and ambient medium, characteristics of the nozzle and rheology of the aqueous nanoparticle suspension directly impacts the droplet size distribution and hence dispersion of nanoparticles.

Monday, November 25, 2019 1:45PM - 3:16PM – Session L38 Porous Media Flow General II 620 - Prabir Daripa, Texas AM University
1:45PM L38.00001 Microscale heterogeneous pore occupancy with variable background pressure gradient. OLIVER MCRAE, Boston University, T.S. RAMAKRISHNAN, Schlumberger-Doll Research, JAMES BIRD, Boston University — Fluid flow through small length scale networks—porous rock, or tumor vasculature—is typically governed by a pressure driven flow dominated by viscous resistance. In the displacement of an immiscible nonwetting fluid, in addition to an external pressure, an internal capillary pressure at the interface of the fluids causes spontaneous movement of the fluid interface. This creates two characteristic regimes: one dominated by capillarity (imbibition), and one dominated by viscosity (drainage). However, the role of the background pressure gradient and heterogeneity on microscale displacement dynamics, and subsequent pore occupancy is unclear. Here we show that the interaction between two parallel pores and a common node with a background resistance is the simplest system to exhibit two distinct imbibition regimes. With this pore doublet model and numerical simulations, we uncover a crossover in pore occupancy as a function of channel size ratio and outlet resistance, and remarkably obtain the same relationship between capillary number and the global residual fluid saturation previously observed in sandstone cores.

1:58PM L38.00002 Study of Flow Over and Through the Porous Bed: Experimental and Numerical Study, NARENDRA KUMAR PATEL, JUNKIE GUO, DAVID ADMIRAAL, University of Nebraska Lincoln — Guo et al. (2016) suggested velocity profile in vegetated flows. Objectives of this research are to extend the velocity profile suggested by Guo et al. (2016) to generate velocity profile over and through the porous gravel bed, and suggest values for the fitting parameters in equations. Numerical-Stokes-Forchheimer equation for flow within the bed and Navier-Stokes equation for flow above the bed are solved simultaneously to generate unified velocity profile. Experimentally, dye was injected in the gravel bed at different depths and its peak concentration was observed at multiple locations at downstream side. A fiber optics based sensor was developed and time for peak concentration at different locations was identified by LabVIEW to generate velocity profile in bed. An ADV is used to measure flow velocity above the bed. V-notch is used to measure total flow at the end of flume. Numerically, Fluent software is used to simulate combine flow field in and above the porous bed. Our preliminary experimental data well matched with numerical results and help us to find values of fitting parameters in theoretical equations. Reference: Junkie, Guo; Jianmin Zhang (2016) Velocity distributions in laminar and turbulent vegetated flows. J. of hydraulic research, volume 54, issue 2, pp 117-130.

2:11PM L38.00003 Gravity-driven sliding motion on soft porous layer.1, RUNGUN NATHAN, Penn State Berks, ZENGHAO ZHU, QIANHONG WU, Villanova University — Soft porous lubrication is a new concept in porous media flow. In this paper, we report a novel experimental study to investigate the gravity-driven sliding motion of a planar board over a tilted soft porous layer. A laser displacement sensor was used to measure the motion of the board, while a high-speed camera was adopted to capture the detailed compression of the porous layer when the board glided over it. The pore pressure generation, as a result of the compression, was recorded by pressure sensors mounted on the bottom surface of the porous layer. One finds that, the pressure distribution agrees well with theory developed by Wu & Sun (Wu Q. & Sun Q., Med. Sci. Sport. Exerc. 2011, 43:1955-63). Extensive parametric study was performed by varying the center of gravity of the planar board, the tilted angle and the porous material. Consistent agreement between the theory and experimental results was obtained. It shows that, the effect of soft porous lubrication is enhanced when the center of gravity moves toward to the trailing edge of the planar board, or the slope of the porous layer is increased, or smoother fibrous surface is used.

2:24PM L38.00004 Experimental study on the phase behavior of fluids confined in nanoporous media.1, XINGDONG QIU, Department of Petroleum Engineering, University of Wyoming, 1000 E. University Avenue, Laramie, WY 82071-2000, USA, SUGATA P. TAN, Planetary Science Institute, Tucson, Arizona 85719-2395, USA, MORTEZA DEJAM, HERTANTO ADIHARMA, Department of Petroleum Engineering, University of Wyoming, 1000 E. University Avenue, Laramie, WY 82071-2000, USA — Phase behavior of fluids in nanoporous media is of great significance in science and engineering applications and has direct and crucial implications in disciplines like drug delivery, CO2 sequestration, and hydrocarbon production and recovery enhancement from unconventional reservoirs, e.g., tight shale formations. It has been well-known that confinement can play an essential role on the abnormal physical behavior of nano-scaled fluids, compared to their counterparts in bulk space. Recently, we developed a new isochoric method using a high-pressure Differential Scanning Calorimeter (DSC) to measure the phase transition of both pure substances and mixtures confined in nanoporous media (SBA-15 matrices with different pore diameters), which turns out to be quite straightforward and reliable. Particularly, this method allows us to achieve detailed insights with regard to the criticality of confined fluids as well as the heat involved during capillary condensation, both of which have hardly or even never been experimentally explored, especially with confined mixtures. The results demonstrate that confinement can shift the critical point of fluid and have an appreciable effect on the heat released during phase transition.

2:37PM L38.00005 A study of the effect of shear thinning in EOR by surfactant-polymer Flooding1, PRABIR DARIPA, Department of Mathematics, Texas AM University, ROHIT MISHRA, Department of Mechanical Engineering, Texas AM University — We have developed an in-house robust hybrid method for multi-component multi-phase flow in porous media arising in the context of enhanced oil recovery. This hybrid method is based on modified method of characteristics and discontinuous finite element methods. We use this method to study the effects of a mixing model and effect of shear thinning on the fingering in enhanced oil recovery by surfactant-polymer flooding. This is an ongoing project and results of our study will be presented.

2:50PM L38.00006 Optimization study of porous wind fence on reducing and stabilizing fluctuating pressure in the wake region. XINGZHOU ZHOU, HEECHANG LIM, Pusan National University — The sheltering effect of the porous wind fence on wind flow has been highlighted last several decades, which provides a tremendous reduction and stability of wind speed in vegetation area by changing the porosity. This study aims to examine a variety of the wind fence, which varies the porosity (i.e., 0.1, 0.2, 0.3, 0.35, 0.4, 0.45, 0.5, 0.7) and the location of a porous fence placed in a simulated turbulent boundary layer. The sheltering effect was observed by the mean and fluctuating quantities such as velocity and pressure variation in the wake of porous fences. The study performed a numerical simulation by using a 2-equation RANS model such as k- turbulence closure models, k- SST, and LES models. The study analyzes wind and pressure characteristics behind wind fences under flat smooth surface as well as rough. In a preliminary result, the wind speed behind the wind fence decreased more than 50% in the porosity 0.1-0.6, which is considered as the wind-protect (i.e., stable) area. In addition, the numerical predictions show good agreements with the existing experiments. Regarding optimum porosity, around 0.3-0.5 seems to be most effective in terms of reduction in wind speed and fluctuating pressure in the wake of wind fence.
which fluid is entrained is found to be dependent on the drop Froude number. Significant retention only occurred for \( Fr < \) of the trailing column of entrained fluid. These dynamics are independent of the far-field nature of the droplets wake. This timescale over a droplet is significantly slowed by its interaction with the ambient stratification over a characteristic timescale, which coincides with the decay of gradients. The present work aims to quantify and explain retention of an oil droplet rising through a transition between two homogeneous-density fluids. Our estimated approximately 40 percent of that oil was trapped beneath the ocean surface, primarily in regions with strong oceanic density gradients. The authors gratefully acknowledge funding from the Hellman Faculty Fellows Fund.

1EPSRC EP/P009751/1 and ERC H2020 805469

Monday, November 25, 2019 1:45PM - 3:03PM –
Session L39 Geophysical Fluid Dynamics Stratified Flow II

1:45PM L39.00001 Nonlinear dynamics of destabilized array of vortices in stratified fluid1, MATTHIEU MERCIER, CNRS-IMFT, UMR5502, FRANCE — The settling dynamics of small objects in stratified fluids is important to understand the fate of the biomass in lakes or oceanic environments, for industrial applications such as waste-water disposal. More specifically, the settling of a suspension of solid particles is a fundamental problem, well-studied for a homogeneous fluid and barely investigated numerically for stratified fluids. We present experimental results on the settling dynamics at low Reynolds number of an initially homogeneous suspension of non-Brownian particles immersed in a linearly stratified viscous fluid, due to a linear variation with depth of salt. We characterize the mean and fluctuations of these quantities for various stratification intensities, in order to quantify the influence of the stratification on settling. We compare these results with similar experiments realized in a homogeneous viscous fluid.

1JSPS Kakenhi 17K05561

1:58PM L39.00002 Low-Reynolds dynamics of a suspension of spheres in a stratified fluid1, MESKO WALEN, J. Fluid Mech. 310, (1996). The authors gratefully acknowledge funding from the Hellman Faculty Fellows Fund.

2:24PM L39.00004 Retention of oil droplets rising in a stratified fluid: Part 2. Dynamics1, MAXIME THEILLARD, DUSTIN KLECKNER, SHILPA KHATRI, University of California, Merced — During the Deepwater Horizon oil spill in 2010, about 5 million barrels of petroleum were discharged from the Macondo Well into the Gulf of Mexico. Oceanographic studies (McNutt, 2012) estimated approximately 40 percent of that oil was trapped beneath the ocean surface, primarily in regions with strong oceanic density gradients. The present work aims to quantify and explain retention of an oil droplet rising through a transition between two homogeneous-density fluids. Using laboratory experiments, we examined droplet behavior for a range of droplet densities, drop sizes, and ambient stratification profiles. We observed that droplets experienced significant slow down as it passed through the stratification. We characterized the droplet slowdown by delineating two droplet motion timescales: entrainment time, the span of time droplet velocity was less than the upper layer terminal velocity, and retention time, the span of time the droplet was retained in the transition layer. We observed a strong relationship between the two metrics, where retention time is a function of the length of time that dense fluid is entrained and the magnitude of the droplet’s slowdown.

1Hellman Faculty Fellows Fund

The authors gratefully acknowledge funding from the Hellman Faculty Fellows Fund.

1The authors gratefully acknowledge funding from the Hellman Faculty Fellows Fund.
2:37PM L39.00005 Cluster formation and self-assembly in stratified fluids: a novel mechanism for particulate aggregation1, RICHARD MCLAUGHLIN, ROBERTO CAMASSA, University of North Carolina at Chapel Hill, DANIEL HARRIS, Brown, ROBERT HUNT, University of North Carolina at Chapel Hill, ZELIHA KILIC, Arizona State University — We observe experimentally and mathematically model a new fundamental attractive mechanism we have found in our laboratory by which particles suspended within stratification may self-assemble and form large aggregates without need for short range binding effects (adhesion). This phenomenon arises through a complex interplay involving solute diffusion, impermeable boundaries, and the geometry of the aggregate, which produces toroidal flows. We show that these flows yield attractive horizontal forces between particles. The collective motion we observed experimentally appears to solve jigsaw-like puzzles on its way to organizing into a large scale disc-like shape, with the effective force increasing as the collective disc radius grows. Control experiments with two objects (spheres and oblate spheroids) isolate the individual dynamics, which are quantitatively predicted through numerical integration of the underlying equations of motion. With this two-body information, we present simulations with hundreds of spheres which reproduce many of the features of our self-assembly experiments.

1DMS-1910824, DMS-1909521, ONR N00014-18-1-2490.

2:50PM L39.00006 Cluster formation and self-assembly in stratified fluids: Particle imaging velocimetry and modified Stokesian dynamics® ROBERT HUNT, ROBERTO CAMASSA, University of North Carolina at Chapel Hill, DANIEL HARRIS, Brown, ZELIHA KILIC, Arizona State University, RICHARD MCLAUGHLIN, University of North Carolina at Chapel Hill — We report on a new fundamental attractive mechanism we have found at the UNC Joint Fluids laboratory by which particles suspended within stratification may self-assemble and form large aggregates without need for short range binding effects (adhesion). This phenomenon arises through a complex interplay involving solute diffusion, impermeable boundaries, and the geometry of the aggregate, which produces nontrivial fluid flows. Numerical simulations are directly compared with particle imaging velocimetry for a single oblate spheroid and are shown to agree both qualitatively and quantitatively with PIV data. Numerical simulations with two spheres at an observation distance dictated by the calculations of an effective force between pairs of particles. With this two-body information, we extend to multiple bodies by modifying a Stokesian dynamics solver to include these forces. Simulations with hundreds of spheres reproduce many of the features of our self-assembly experiment.

1DMS-1910824, DMS-1909521, ONR N00014-18-1-2490.

Monday, November 25, 2019 1:45PM - 3:16PM
Session L40 Jets: Control 6b - Peter Schmid, Imperial College of London

1:45PM L40.00001 Determining spray axial velocity from focused-beam X-ray radiography. JULIE BOTHELL, TIMOTHY MORGAN, Iowa State University, ALAN KASTENGREN, Argonne National Laboratory, THEODORE HEINDEL, Iowa State University, ENERGY SYSTEMS DIVISION, X-RAY SCIENCE DIVISION TEAM, EXPERIMENTAL MULTIPHASE FLOW LAB TEAM — Coaxial atomizing sprays are used across a variety of industries from gas turbines to food processing. Far-field spray dynamics depend on the primary breakup region, but this region is challenging to study as it contains thick liquid that is opaque to visible light. However, X-rays are capable of penetrating the dense liquid region, providing insight that is unavailable from visible light testing methods. This study modifies a method used in previous studies for determining the mass-average axial velocity from a diesel injector spray, and applies it to investigate the mass-average axial velocity from a coaxial atomizing spray. The original method was developed for a narrow-angle, time-varying spray, but was modified in this study for a wide-angle, steady-state spray. Experiments at the Advanced Photon Source at Argonne National Laboratory provided focused-beam X-ray radiographs along the spray. The mass-average axial velocity along the spray increased linearly with axial distance from the nozzle for varying momentum ratios. The slope of the velocity-distance relation also increased linearly when plotted as a function of gas flow rate.

1This work was sponsored by the Office of Naval Research under grant number N00014-16-1-2617. Portions of this work was performed at the 7-BM beamline of the Advanced Photon Source, a U.S. Department of Energy (DOE) Office of Science User Facility operated for the DOE Office of Science by Argonne National Laboratory under Contract No. DE-AC02-06CH11357.

1:58PM L40.00002 The azimuthal and wavy deformation of vortex rings and the formation of separated flows in a round jet using synthetic jets. AKINORI MURA MATSU, College of Science and Technology, Nihon University, KOHEI TANAKA, Graduate School of Science and Technology, Nihon University — Vortex rings are periodically formed in the initial region of a round jet and change to a wavy shape in the azimuthal direction. As a result, the vortex rings collapse three-dimensionally. It is suggested that the azimuthal deformation of vortex rings is affected by streamwise vortices. In order to investigate the relation between the azimuthal deformation of the vortex ring and the streamwise vortices, we attempted to artificially deform the vortex ring in the azimuthal direction using synthetic jets. The synthetic jets are formed utilizing a sound wave with a natural frequency for the vortex-ring formation. The vortex rings azimuthally and wavy deform by introducing disturbances through small holes at a nozzle exit, so that the streamwise vortices are generated at certain locations in the round jet. At the same time, the jet branches off slanting upward at the initial region in relation to azimuthal deformation of the vortex ring. The streamwise vortices are formed inside the vortex ring and move to the outside of the vortex ring, when the vortex ring is moving the downstream. At this time, the streamwise vortices are separated from the vortex ring. The branched flows, as similar to side jets, are formed by moving the streamwise vortices.

2:11PM L40.00003 Adjoint-based analysis of controllability of turbulent jet noise. SEUNG WHAN CHUNG, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Past efforts have used optimal control theory, based on the numerical solution of the adjoint flow equations, to perturb turbulent jets to reduce radiated sound. These have been successful in that sound is reduced, with consequent changes to large-scale turbulence structures in the flow. However, control remains challenged by the chaotic dynamics of the turbulence, which degrades smoothness of cost functional in control parameter space, thus limiting control effects to a relatively short time horizon. Wave packets can be a useful and relatively deterministic model for the large-scale noise mechanisms in jets. Using discrete-exact, dual-consistent adjoint gradients in conjunction with high-fidelity simulations, we assess the growth of sensitivity in time and general controllability. The most effective immediate control of a $M = 1.3$ jet is achieved with a control-time horizon of $32 AD/a_{\infty}$; longer or shorter time horizons prove less effective, which thus marks a balance between increasing authority through sensitivity and the disruptive influence of chaotic effect in the turbulence for this objective. After the control period, the rate at which the jet returns to its baseline louder state also presents a potentially important time scale for the flow and chaotic dynamics.
2:24PM L40.00004 Dynamics of Axisymmetrically Excited Transverse Jets\textsuperscript{1}, ELIJAH HARRIS, DAVID D. W. REN, ANDREÁ BESNARD\textsuperscript{2}, STEPHEN SCHEIN, ROBERT M’CLOSKLEY, ANN KARAGOZIAN, University of California, Los Angeles, LUCA CORTELEZZI, Politecnico di Milano — The present experimental study investigates axisymmetric excitation of a gaseous jet issuing into a uniform crossflow as pertains to jet dynamics as well as structural and mixing characteristics. A naturally absolutely unstable (AU) transverse jet, with a jet-to-crossflow momentum flux ratio of $J=6$, is forced with a variety of different periodic waveforms including sinusoidal, square wave, and multi-pulse square waves. For specific perturbation amplitudes and within specific forcing frequency regimes, the jet locks-in to the forcing frequency, prior to which there is evidence of quasi-periodicity. The critical conditions to achieve lock-in differ amongst the various excitation waveforms, where the sinusus forcing cases have the greatest challenges in achieving lock-in for this AU jet. As one increases the forcing amplitude beyond lock-in, the jet displays complex synchronization dynamics and mode shapes, en route to more chaotic behavior, as quantified through snapshot proper orthogonal decomposition (POD) analysis of the velocity field extracted via stereo particle image velocimetry (PIV), and time delay embedding of velocity fluctuations along the jet shear layer.

\textsuperscript{1}Supported by AFOSR Grant FA9550-15-1-0261 (PO: Dr. Gregg Abate) and NSF Grant CBET-1933310 (PD: Dr. Ron Joslin)

\textsuperscript{2}Currenty with Exponent

2:37PM L40.00005 Open-loop control of a weakly non-linear swirling jet instability by harmonic forcing, CALUM SKENE, Imperial College London, UBAID QADRI, University of Cambridge, PETER SCHMID, Imperial College London — Highly swirling flows are often prone to a global instability with an azimuthal wavenumber of $m=1$, where the sign is chosen so that the instability precesses against the mean-flow swirl. This instability is linked to the recirculation region typical of a vortex breakdown induced by high swirl, and can affect many types of flow. In particular, in lean premixed combustors swirling flows are often utilised to stabilise a flame onto a burner, as the recirculation region caused by vortex breakdown acts as a natural ‘flame holder’. We investigate the manipulation of the global instability by harmonic forcing through a weakly non-linear analysis of an incompressible swirling jet. The flow is expanded about the critical Reynolds number where instability first occurs. Linear effects are obtained at first order, including both the unstable mode and the response to harmonic forcing. At second order, harmonics and base-flow modifications caused by non-linear interactions of the mode and forced response are obtained. An equation for the mode amplitude is obtained at third order, allowing for the effect of harmonic forcing to be quantified. In this manner, the effectiveness of control across a variety of forcing frequencies and azimuthal wavenumbers is assessed.

2:50PM L40.00006 Internal Flow Dynamics and Spray Characteristics in Liquid Swirl Injectors, JACOB GAMERTSFELDER, PRASHANT KHARE, University of Cincinnati — The focus of this research effort is to investigate the internal flow dynamics and subsequent spray characteristics in swirling liquid fuel injectors. While significant progress has been made in the past to enhance the understanding of the internal dynamics of the vortex generator and resulting liquid sprays, limited studies exist that capture both the detailed mechanistic processes leading to swirling liquid and the resulting spray formation. This effort will fill this gap in the literature. The theoretical formulation is based on three-dimensional incompressible Navier-Stokes equations with surface tension. A volume of fluid (VOF) method is used for interface capturing. The swirl injector geometry consists of three equally spaced (at 120 degrees) tangential inlets of 1 mm diameter, a 6 mm vortex generator and a 2 mm diameter spray nozzle with a $L/D$ ratio of 12. The operating conditions consists of a Weber number of 1556, corresponding to a flow rate of 25.51 g/s at an injector pressure of 0.579 MPa. The ambient chamber pressure and temperature are 4 MPa and 300 K, respectively. Preliminary results show that the swirl generated in the injector leads to the formation of a hollow cone spray that breaks up to form ligaments and droplets. A systematic analysis will be conducted to quantitatively identify the processes leading to the formation of liquid swirl near the injector walls, air core in the center and subsequent spray formation including droplet size distributions, breakup length and spray angles.

3:03PM L40.00007 Helical Vortex Characterization in Swirling Jets from Planar Measurements\textsuperscript{1}, BENJAMIN EMERSON, TIM LIEUWEN, Georgia Institute of Technology — The fluid mechanics of swirling jets are highly three-dimensional, but most state of the art measurement capabilities are only planar. This work demonstrates an experimental data analysis methodology for swirling jets. The methodology is implemented on planar velocity field data to interpret the dynamical, three-dimensional topology of the velocity field. The methodology blends physical understanding of reacting swirling jets with clues that are present in stereoscopic particle image velocimetry data to infer a helical vortex tube inclination angle. With this inclination angle, the planar measurement data can be revolved to re-construct the three-dimensional velocity field.

\textsuperscript{1}This research was partially supported by the Air Force Office of Scientific Research (contract #FA9550-16-1-0442), contract monitor Dr. Chiping Li.

Monday, November 25, 2019 1:45PM - 3:03PM — Session L41 CFD: Discontinuous Galerkin Methods

1:45PM L41.00001 High-order extended discontinuous Galerkin methods for sharp shock-capturing\textsuperscript{2}, MARTIN OBERLACK, MARKUS GEISENHOFER, FLORIAN KUMMER, BJÖRN MUELLER, TU Darmstadt, FLUID DYNAMICS TEAM — We study unsteady high Mach number flows using a Discontinuous Galerkin (DG) solver (Mueller 2016) that was extended to immersed boundary methods (IBM) using level sets. Cell-agglomeration was applied for small and ill-shaped cut cells. To cope with shocks, we used an artificial viscosity shock-capturing approach (Persson 2006) that was coupled to IBM (Geisenhofer 2019). There a shock sensor is used to identify critical cells and artificial viscosity smoothens the solution. The severe time stepping restrictions due to an explicit time step together with the additional diffusive term are dealt by an adaptive local time stepping (LTS) approach (Winters 2014) that dynamically (re-)partitions the grid according to their local time step. Presently we extend the shock-fitting techniques using an eXtended DG (XDG) method (Kummer 2016) to regain the desirable DG convergence rates and a sharp jump representation. For this, we first investigate shock tracking in 1D, where we prescribe the flow properties across shocks using the Rankine-Hugoniot conditions in combination with the level set propagation speed. Second, we consider the formation of stationary shocks in 2D, where the artificial viscosity IBM, LTS method will be the initial solution to be morphed to a XDG shock-fitting approach.

\textsuperscript{2}MG & BM were supported by the GS Excellence CE at TU Darmstadt, FK by DFG CRC 1194/B06 and BM by DFG Grant WA 2610/2-1. Calculations were done on Lichtenberg HPC cluster at TU Darmstadt.
1:58PM L41.00002 Reacting flow simulations using high-order discontinuous Galerkin methods1, KIHIRO BANDO, MATTHIAS IHME, Stanford University, MICHAEL SEKACHEV, Total — High-order discontinuous Galerkin (DG) methods have been an increasingly popular topic of research for enabling high-fidelity simulations on complex geometries. They present several attractive features such as high-order accuracy on arbitrary mesh topologies, compact implementation and support of advanced hp-refinement strategies which can be leveraged to increase computational efficiency. However, the application of such methods to flows presenting complex thermodynamics and chemical reactions is still sparse. In particular, the accurate prediction of such flows requires the consideration of complex transport and the treatment of stiff reaction chemistry. This talk will discuss challenges associated with reacting flow simulations in the context of a DG discretization. Simple one-dimensional cases will first be investigated to gain insight at the fundamental properties of the scheme when applied to such flows. Subsequently, more challenging cases in multiple dimensions will be examined to highlight the performance as well as the need for further developments to enable reacting flow simulations using DG methods.

1Total EP Research and Technology USA, LLC (Award Number: 134818)

2:11PM L41.00003 Development of a Four-Way-Coupled Lagrangian Particle Method for Discontinuous Galerkin Schemes1, ERIC CHING, MATTHIAS IHME, Stanford University — In this talk, we present an Euler-Lagrange methodology within the framework of discontinuous Galerkin methods. We discuss strategies to track particle trajectories through curved elements near walls while taking into account finite particle sizes. The back-coupling of particles to the carrier gas is treated in an efficient manner. In addition, we introduce a particle-particle collision algorithm that utilizes information provided by the geometric mapping from physical space to reference space. In doing so, the proposed algorithm reduces the number of particle pair inspections compared to standard methods. The algorithm is verified by drawing comparisons to kinetic theory. To demonstrate the capability of the multiphase flow solver, we consider a variety of test cases that include hypersonic dusty flows over blunt bodies and erosive action of sand blasting.

1This work was supported by a Stanford Graduate Fellowship and an Early Career Faculty grant (NNX15AU58G) from the NASA Space Technology Research Grants Program.

2:24PM L41.00004 An Enriched-Basis Discontinuous Galerkin Method for Wall-Modeled Large-Eddy Simulations1, STEVEN R. BRILL, MATTHIAS IHME, Stanford University — We developed an enriched-basis discontinuous Galerkin (DG) method for wall-modeled large-eddy simulations (WMLES) of turbulent flows. Ordinarily, higher-order methods, such as DG-schemes, require significant mesh refinement near the wall in order to resolve the turbulent boundary layer without spurious oscillations due to large gradients in the boundary layer. To avoid this issue, we enrich the traditional polynomial basis with a problem-specific non-polynomial basis function in the near-wall elements in order to capture the inner boundary-layer structure while still using a coarse mesh. In this method, the enrichment function is not required to be active, but is chosen by the Galerkin procedure when it is optimal, such as near the wall. As a result, the enrichment basis functions capture the mean behavior near the wall, the polynomial basis functions resolve the large eddies, and a subgrid scale model represents the small scale behavior. We discuss the procedure for choosing the proper enrichment functions for a problem and integrating non-polynomial basis functions into the DG framework. The method is demonstrated in application to turbulent channel flows and other canonical wall-bounded turbulent flows. 

1Financial support from the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program and the NASA Early Career faculty program are gratefully acknowledged.

2:37PM L41.00005 A High-Order Lagrangian Discontinuous Galerkin Method on Hybrid Curved Meshes1, XIAODONG LIU, NATHANIEL MORGAN, DONALD BURTON, Los Alamos National Lab, HIGHORDER LAGRANGIAN AT LANL TEAM — We present a high-order (up to fourth order) Lagrangian DG hydrodynamic method using subcell mesh stabilization (SMS) for compressible flows on hybrid curved meshes (up to cubic meshes) in 2D Cartesian coordinates. The physical evolution equations for the specific volume, velocity, and specific total energy are discretized using a modal DG method. The challenge for Lagrangian hydrodynamics is to guarantee the stable mesh motion for the edge vertex, that can easily deform in unphysical ways. With SMS, each cell is decomposed into several subcells, that move in a Lagrangian manner. The edge vertex is surrounded by several subcells so that enough information can be obtained, that is similar to the corner vertex. As such, the same multidirectional approximate Riemann problem can be solved for these both types of vertices. This SMS scheme enables stable mesh motion and accurate solutions in a context of the Lagrangian high-order DG method. We also present effective limiting strategies that ensure monotonicity of the primitive variables with the high-order DG method. A suite of test problems are calculated to demonstrate the designed order of accuracy of this method and the robustness for the strong shock problems.

1We gratefully acknowledge the support of the NNSA through the Laboratory Directed Research and Development (LDRD) program at Los Alamos National Laboratory. The Los Alamos unlimited release number is LA-UR-19-27300

2:50PM L41.00006 A Recovery-Assisted Discontinuous Galerkin Method for the Compressible Navier-Stokes Equations1, ERIC JOHNSON, PHILIP E. JOHNSON, University of Michigan — The Discontinuous Galerkin (DG) method is a promising approach for high-fidelity simulations of turbulent flows in complex geometries, given the method’s capability for arbitrarily high orders of accuracy on unstructured meshes. We propose a DG approach for advection-diffusion equations (such as the Navier-Stokes equations) based on the principle of Recovery, whereby the underlying solution between two adjacent computational elements is “recovered” into a smooth, more accurate polynomial representation. By combining a biased approach to Recovery for advection with a compact gradient recovery for diffusion, we achieve $2p + 2$ order of accuracy in the cell-average error on a nearest-neighbors stencil, where $p$ is the degree of the polynomial basis, and demonstrate higher efficiency than other state-of-the-art DG approaches. Several test cases, including compressible turbulence, are presented to support our method.

1Computing resources were provided by the NSF via grant 1531752 MRI: Acquisition of Conflux, a Novel Platform for Data-Driven Computational Physics (Tech. Monitor: Ed Walker) and by the Office of Naval Research, Grant N00014-16-1-2032 (under Dr. Ki-Han Kim).
in the streamwise direction while it becomes a U-shaped curve when the flow is super-critical. Time-dependency effects are also investigated.

Interestingly, when the flow is sub-critical the optimal wind turbine thrust set-point assumes a sinusoidal behavior of the wind turbines. We implement an optimization model in order to derive the turbine thrust coefficient distribution that maximizes the atmosphere in three layers; the wind-farm drag force is applied over the whole wind-farm area and is directly proportional to the thrust set-point and in the free atmosphere. These waves impose significant pressure gradients in the boundary layer causing detrimental consequences in terms of high-gradient surface slope on the energy entrainment and the vertical transport of momentum and kinetic energy over the model wind farm.

In order to better understand the effect of the complex terrains, wind tunnel experiments were performed on a 1:290 scale model wind farm. To explore the advantages of complex topographic regions, scaled-down turbine in the model of Ollantaytambo region of Peru was used to analyze the wake effects through power output and particle image velocimetry (PIV). The results from the model wind farm over the complex terrain are indicative of greater efficiency in wind farms in terms of power output. Additionally, we will discuss the role of high-gradient surface slope on the energy entrainment and the vertical transport of momentum and kinetic energy over the model wind farm.

2:24PM L42.00004 Effect of nocturnal low-level jet on wind farm performance. SRINIDHI NAGARADA GADDE, RICHARD STEVENS, University of Twente — Large-eddy simulations of wind farms in a quasi-stationary, nocturnal, stable boundary layer are performed to understand the effects of the low-level jets (LLJ) on the farm performance. The effect of the LLJ is studied by systematically varying the cooling rate at the surface. The height of the boundary layer decreases with the increasing cooling rate and forms a LLJ due to the frictional decoupling at the inversion layer. We find that the power production of the wind farm increases with the cooling due to the high shear in the LLJ. For strong cooling, the destruction of turbulence by buoyancy causes a drop in the vertical entrainment; consequently, the turbines in the rear of the wind farm produce less power compared to the turbines operating in a low or moderately stable boundary layers. In addition to the power production, we analyse the flow structures to better understand the wind farm-boundary layer interaction in the highly stratified atmosphere.

2:37PM L42.00005 Positive effect of a synthetic low-level jet on the mean power and momentum transport of a wind-turbine array. ALI DOOSTTALAB, DIEGO SIGUENZA, JOSSY O’DONNELL, Purdue University, WALTER GUTIERREZ, Texas Tech University, VENKATESH PULLETIKURTHI, Purdue University, YAOQING JIN, University of Texas at Dallas, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign, LUICANO CASTILLO, Purdue University — Nocturnal low-level jet (LLJ) is a distinctive phenomenon at the top of stable boundary layers. A low-level velocity peak results in attractive power resource for wind turbines. However, a maximum in the mean wind speed profile implies the co-existence of positive and negative mean shear in the vicinity of the peak. To gain better understanding of the impact of LLJ (e.g., positive & negative shear) on the performance of a scaled-down power plant, with similar LLJ profile in the atmosphere were performed using particle imaging velocimetry (PIV). The results from single turbine and 3 x 2 turbines array in the positive and negative shear regions of the synthetic LLJ were compared to canonical turbulent boundary layer. For both positive and negative shear cases a 12% increase in power generation in the 2nd row were observed in comparison to the unstable boundary layer condition. In addition, the role of energy entrainment induced by the positive and negative shear of the LLJ and the contribution to the vertical transport of momentum and kinetic energy across the turbine array are explored in depth.

2:50PM L42.00006 The low-level jet role on the mean power and momentum transport of vertical axis wind turbines. DIEGO SIGUENZA, ALI DOOSTTALAB, HUMBERTO BOCANEGRA-EVANS, Purdue University, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign, LUICANO CASTILLO, Purdue University — The stable temperature stratification at the lower part of the atmosphere causes a particular phenomenon known as low-level jet (LLJ) where its velocity peak results in attractive power resource for wind turbines. The positive and negative shear layers of the LLJ influences the wake recovery by encouraging the energy entrainment into the horizontal-axis wind farm canopy. To extend the knowledge of LLJ impact on wind farms, we aim to explore the effect of the LLJ on vertical-axis wind turbines (VAVT) in terms of its wake energy entrainment. The LLJ was synthetically generated in a wind tunnel to test different arrays of scaled-down VAWT where we measured the velocity fields downstream particle image velocimetry (PIV), and the power output of the turbines. We will discuss the role of the wake energy entrainment induced by the positive and negative shear in comparison with a regular unstable boundary layer scenario.

Monday, November 25, 2019 3:20PM - 4:13PM –
Session M01 Flash Oral Presentations: Turbulence 6a - Mark Glauser, Syracuse University
Subspaces be compared to a reduced-order model constructed using static (i.e. time invariant) POD modes. Reduced-order modeling of: (1) transient instabilities in Kuramoto-Sivashinsky equation, and (2) transient flow over a bump. The results will be validated using deterministic/stochastic systems constructed by projection of the full-dimensional dynamics onto a time-dependent basis. To this end, we explore the possibility of a deep learning model to mimic the remap function. Our deep neural network is trained on the nodal velocities of a structured mesh before and after the remap step in our Eulerian hydrocode. We show that our deep learning model can accurately reproduce the remap function to acceptable accuracy, and we compare speeds between our DNN and the existing remap function. This work continues to demonstrate that deep learning models can enhance numerical predictive capabilities. Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-NA0003525.

This study was provided funding via research grants by the Air Force Office of Scientific Research (AFOSR) grant number FA9550-19-1-0081 (Program Manager: Dr. Greg Abate) and Spectral Energies LLC.

Towards a machine learning method for simulating turbulence-shockwave interactions. In recent years, machine learning has been used to create data-driven solutions to problems for which an algorithmic solution is intractable, as well as fine tuning existing algorithms. This research applies machine learning to the development of an improved finite-volume method for turbulence-shockwave interactions. Shock capturing methods make use of complicated nonlinear functions that are not guaranteed to be optimal. Because data can be used to learn complicated nonlinear relationships, we train a neural network to improve the results of WENO5. We also post-process the outputs of the neural network to guarantee that the method is consistent. The training data is generated using integrable functions that represent the waveforms we would expect to see while simulating a PDE, which gives us an exact mapping between cell averages and interpolated values. We demonstrate our method on linear advection of a discontinuous function, the inviscid Burgers equation, and the 1D Euler equations. Specifically, we examine the Shu-Osher problem, a toy problem for turbulence-shockwave interactions. We also show preliminary results for dynamically trained models.

This research was funded by the National Natural Science Foundation of China (Grant No. 51776011).

Reinforcement learning enabled control of chaotic dynamics. A deep RL policy network, based on proximal policy optimization, is employed to stabilize the unstable fixed points and periodic orbits embedded in the chaotic-attractor of the Lorenz system. Previous attempts to control the underlying chaotic trajectories have relied on linearization of the dynamics around the targeted solutions, or on time-delayed feedback based on the output variables. However, there are certain caveats associated with these control approaches such as, requiring an a priori understanding of the chaotic system to be controlled, and difficulty in stabilizing certain classes of periodic orbits. These issues are overcome in the RL enabled control, especially when long term correlations are accounted for using layers of Long Short Term Memory (LSTM) cells in the policy network.

This work continues to demonstrate that deep learning models can enhance numerical predictive capabilities. Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-NA0003525.

This research was funded by the National Natural Science Foundation of China (Grant No. 51776011).

Real-Time Reduced Order Modeling Using Time Dependent Subspaces. We present real-time reduced-order models for deterministic/stochastic systems constructed by projection of the full-dimensional dynamics onto a time-dependent basis. To this end, we leverage a scalable algorithm to extract time dependent modes from highly transient data sets. We will present two case studies for the reduced-order modeling of: (1) transient instabilities in Kuramoto-Sivashinsky equation, and (2) transient flow over a bump. The results will be compared to a reduced-order model constructed using static (i.e. time invariant) POD modes.

NASA Grant 80NSSC18M0150

POD analysis of the unsteady behavior of the wake under the influence of laminar to turbulent transition in a compressor cascade. For low incidence angle and Reynolds number, the transition region is usually located closer to the trailing edge, which leads to a strong interaction of the vortex shedding of the laminar separation bubble and the wake flow. In this paper, particle image velocimetry (PIV) measurements have been performed in order to analyze the unsteady flow field in a compressor cascade. The instantaneous snapshots have been post-processed by means of proper orthogonal decomposition (POD). The first mode pair allows the instantaneous snapshot to be correctly sorted in the wake field. For all the time steps, according to the extent of the deviation between the time-averaged flow field and the phase-averaged counterpart, the flow field can be divided into three regions associated with the LSB vortex shedding, the wake flow and the interaction between the former two dynamics. The triple decomposition of the velocity fluctuations enables the quantification of the contribution associated with the three coherent motions as well as the stochastic motions to the overall velocity fluctuations.

This research was funded by the National Natural Science Foundation of China (Grant No. 51776011).
3:27PM M01.00008 Sparse Identification of Nonlinear Dynamics in 2D Chaotic Electrohydrodynamic Convection, IGOR NOVOSSELOV, YIFEI GUAN, STEVEN BRUNTON, University of Washington — This study focuses on developing a reduced-order model for the chaotic electro-hydrodynamic (EHD) convection flow between two parallel electrodes with unipolar charge injection. A Lattice Boltzmann Method with two-relaxation times solver [1] is used to obtain a numerical data set for electroconvective instabilities. Under strong charge injection and high electrical Rayleigh number, the system transitions from structured electroconvective vortices to chaotic motion. The chaos in this system is related to the standard Lorenz model obtained from Rayleigh-Benard convection, although this model system exhibits a more complex three-way coupling between the fluid, the charge density, and the electric field. Fluid coherent structures are extracted from the temporally and spatially resolved chaotic system via proper orthogonal decomposition (POD). A nonlinear model is developed for the chaotic time series of these POD coefficients with the sparse identification of nonlinear dynamics (SINDy) algorithm [2]. The resulting sparse nonlinear model captures a similar phase portrait and the dominant chaotic dynamics of the original system. References: [1] Y. Guan and I. Novosselov, Two Relaxation Time Lattice Boltzmann Method Coupled to Fast Fourier Transform Poisson Solver: Application to Electroconvective Flow, Journal of Computational Physics (accepted for publication) (2019). [2] S. L. Brunton, J. L. Proctor, and J. N. Kutz, Discovering governing equations from data by sparse identification of nonlinear dynamical systems, Proceedings of the National Academy of Sciences 113, 3932 (2016).

1This work is supported by the FLAGSHIP2020, MEXT within the Post-K Priority Issue 4 (Advancement of meteorological and global environmental predictions utilizing observational “Big Data”).

3:28PM M01.00009 Wavelet-based Data Compression for Three-dimensional Fluid Flow Simulations on Regular Grids, DMITRY KOLOMENSKYI, RYO ONISHI, HITOSHI UEHARA, Japan Agency for Marine-Earth Science and Technology — High-performance computational fluid dynamics can produce large volumes of output data. Even though data reduction may be performed during the course of computation to store only the quantities of interest such as statistical moments, it is often necessary to store full three-dimensional fields for purposes including simulation restart, time-resolved visualization and exploratory analyses. In this talk, I will present a wavelet-based method for compression of numerical simulation data on regular structured grids. It is inspired by image compression, and it consists of discrete wavelet transform, quantization adapted for floating-point data, and entropy coding. I will discuss different aspects of these numerical techniques, present an open-source software WaveRange, and show example numerical tests, ranging from idealized configurations to realistic global weather simulation data.

3:29PM M01.00010 Data Compression for Turbulence Databases Using Spatio-Temporal Sub-Sampling and Local Re-simulation, CHARLES MENEVEAU, TAMER ZAKI, ZHAO WU, Johns Hopkins University — The size of datasets from numerical simulations of turbulent flows has been growing roughly at the rate expected from Moore’s law. However, storing the rapidly growing amounts of numerical simulation data is very challenging. Motivated by specific data and accuracy requirements for building numerical databases of turbulent flows, data compression using spatio-temporal sub-sampling and local re-simulation is proposed. The DNS data are aggressively subsampled in space and time during the simulation, and a local re-simulation is performed later when the data are needed. Numerical re-simulation experiments for decaying isotropic turbulence based on sub-sampled data are undertaken. The results and error analyses are used to establish parameter choices for sufficiently accurate sub-sampling and sub-domain re-simulation. It is found that the numerical algorithms of the re-simulation have to match the original simulation exactly to reproduce error-free results: any mismatch between the two is found to lead to surprisingly large errors. By carefully controlling the re-simulation parameters, re-simulation errors close to single-precision machine accuracy can be achieved under stringent conditions.

1National Science Foundation (grant # OCE-1633124). Computations were made possible by the Maryland Advanced Research Computing Center (MARCC).

3:30PM M01.00011 Temporal evolution and statistical characteristics of uniform momentum zones using data-informed resolvent hierarchies, ANGELIKI LASKARI, BEVERLEY MCKEON, California Institute of Technology — Experimental data from time-resolved planar particle image velocimetry in streamwise-wall-normal planes of a turbulent boundary layer are used for the determination of temporal evolution and statistical characteristics of uniform momentum zones. Specifically, the temporal variation of both the probability density function (pdf) of the streamwise velocity and the instantaneous number of zones is assessed. Statistically important patterns observed in the experimental data are then used to guide the selection of modes on a self-similar resolvent hierarchy. Although the full range observed for the instantaneous number of zones is not recovered with the use of this restricted number of modes, it is shown that the largest variation observed is due to the modes located in the middle of the logarithmic region. Additionally, results indicate that a single resolvent hierarchy can reproduce a prominent semi-periodic behaviour observed in the experimentally constructed temporal pdf of the streamwise velocity. It is further shown that this behaviour is directly related to the wavenumber of the modes closest to the edge of the logarithmic region.

1The support of ONR under grant number N00014-17-1-3022 is gratefully acknowledged

3:31PM M01.00012 Resolvent analysis-based models of nonlinear Taylor vortex flow, BENEDIKT BARTHEL, California Institute of Technology, XIAOJUE ZHU, Center of Mathematical Sciences and Applications, and School of Engineering and Applied Sciences, Harvard University, BEVERLEY MCKEON, California Institute of Technology — Taylor vortices have recently garnered renewed attention as an analogue for the self-sustaining process (SSP) in wall bounded shear flows. An instability of the mean flow leads to streamwise rolls which form streaks, which in turn become unstable and feedback to the rolls. We seek to shed light on the SSP by first understanding the dominant nonlinear interactions which sustain Taylor vortices. Here we use the resolvent formulation of McKeon and Sharma to treat the nonlinearity not as an inherent part of the governing equations but rather as a triadic constraint which must be satisfied by the model solution. We exploit the low rank linear dynamics of the system to calculate an efficient basis for our solution, the coefficients of which are then calculated through an optimization problem where the cost function to be minimized is the triadic consistency of the solution with itself as well as with the input mean flow. We compare our results to DNS of Taylor Couette flow for a range of Reynolds numbers (Zhu et al., Comp. Phys. Comm., 2018). Our model solution replicates both the flow structures as well as the turbulent statistics observed in the simulations.

1ONR grant number: N0014171-3022
3:32PM M01.00013 A Time-Spectral Analysis for Summed Linear and Product Signals with Applications to Fluid Mechanics and Chaotic Systems1, CHIEN C. CHANG, Guangxi University and National Taiwan University, SHENG-SHENG LIU, YEN-LIANG LEE, Guangxi University, JEN-JEN LIN, Ming-Chuan University — There are time signals of general interest put in the form: Time signal = trend with time + periodic components + residual or randomness. It is of great importance to identify the periodic components whose frequencies and amplitudes may be varying with time. In the past, we have seen excellent works on time-frequency analysis of a signal such as short-time Fourier, wavelet, Hilbert-Huang transforms among others. Yet there are still critical and fundamental issues to be addressed. Notably all the previous analysis (tacitly) assumes that the signal concerned is a linear superposition of its decomposed components no matter whether a base set of functions or no base is employed. Moreover, a common query is that the signal may often be over-decomposed that the analysis with respect to individual modes does not catch the essential features of the signals spectral content. In this study, we propose to develop a principal frequency analysis suitable for general summed linear signals and product signals (beats/wave-packets). As an illustration, this approach of analysis is first applied to several basic examples and then to time-dependent drag in fluid mechanics and signals of a chaotic Rossler system.

1This work was supported by NNSF (China, 11672077) and MOST (Taiwan, 105-2221-E-002 -105 -MY3).

3:33PM M01.00014 Bio-inspired flows in unsteady environments. Part I: highly unsteady ambient flows, MEILIN YU, NARESH POUDEL, University of Maryland, Baltimore County, JOHN HRYNUK, U.S. Army Research Laboratory — Autonomous underwater vehicles (AUVs) and unmanned aerial vehicles (UAVs) usually need to carry out tasks in unstructured and dynamic flow environments. This poses a number of challenges that cannot easily be addressed by approaches developed for highly controlled environments, such as unstructured flows frequently used in experiments and numerical simulation. This work studies the impact of highly unsteady ambient flows on the performance of flapping wings/ fins at relatively high Reynolds numbers (i.e., 12,000 based on the foil chord length). The unsteady flow environment is generated by an array of incline small cylinders or three arrays of staggered ones placed upstream of the flapping wing/fin. A high-order accurate flux reconstruction flow solver with moving/deforming body-fitted unstructured meshes is used to perform the numerical simulation. We find that highly unsteady flow environments dominated by small eddies can always enhance time-averaged thrust generation, no matter how the foil location is changed within the ambient chaotic flows; the effect of environmental unsteadiness on lift production seems to be random. The effects of wing/fin kinematics and the size of cylinders are also studied.

3:34PM M01.00015 Turbulent flow structure associated with interacting 3D bedforms, NATHANIEL BRISTOW, GIANLUCA BLOIS, University of Notre Dame, JAMES BEST, University of Illinois at Urbana-Champaign, KENNETH CHRISTENSEN, University of Notre Dame — Barchan dunes are three-dimensional, crescent-shaped bedforms, and while most commonly associated with aeolian environments, recent observations have shown them to form in subaerial and extraterrestrial environments as well. As barchans migrate in the direction of the flow, they interact with their neighbors, typically by way of a collision. The morphodynamics of such collision processes are complex, where the role of the turbulent flow structure is strongly coupled to that of the sediment transport and morphological change. Here we study the flow structure in a decoupled manner through measurements of the turbulent flow over fixed-bed models of barchan dunes in various configurations involved in a barchan collision process. Particle image velocimetry is used to measure the flow in a refractive-index matched flume environment that enables access to the whole flow field around these geometrically complex bedforms. Presented herein are results from planar PIV measurements in several measurement planes, including the cross-plane, showing the dynamics of turbulent flow structures associated with barchan dunes which are hypothesized to drive the morphodynamics of the dune interaction.

3:35PM M01.00016 Application of the One-Way Navier-Stokes (OWNS) equations to hypersonic boundary layers, OMAR KAMAL, GEORGIOS RIGAS, California Institute of Technology, MATTHEW T. LAKEBRINK, The Boeing Company, TIM COLONIUS, California Institute of Technology — Prediction of linear instability and amplification of disturbances in hypersonic boundary layers is challenging due to the presence and interactions of Tollmien-Schlichting, Mack, and entropic modes. While DNS and global analysis can be used, the large grids required make the computation of optimal transient and forced responses very expensive, particularly when a large parameter space is required. At the same time, parabolized stability equations (PSE) are unreliable for multi-modal interactions. In this work, we instead apply a newly developed technique, the One-Way Navier-Stokes (OWNS) equations, which are based on a rigorous parabolization of the full equations of motion. OWNS removes disturbances with upstream group velocity using a high-order recursive filter. We extend the original algorithm by considering body-fitted curvilinear coordinates incorporating full compressibility and real gas effects. We validate the results by comparison with DNS. We present preliminary results for the optimal growth of disturbances in flat-plate and conical boundary layers. This work has been supported by the Boeing Company through a Strategic Research and Development Relationship Agreement CT-BA-GTA-1.

3:36PM M01.00017 Turbulence drag modulation by combined effect of solid particles injection and synthetic roughness1, CARLOS DUQUE-DAZA, Universidad Nacional de Colombia, JESUS RAMIREZ-PASTRAN, Universidad Santo Tomas — The combined effect of geometric perturbations at one of the walls and injecting spherical solid particles on the behaviour of an incompressible turbulent channel flow at low friction Reynolds number ($Re_f = 180$) was investigated through numerical simulations. The effect of the presence of spherical solid particles was explored from the perspective of the particles-mass-fraction (PMF), whereas spanwise ribs-like and cavity-like geometrical alterations were prescribed as synthetic large scale roughness elements in one of the walls. Values of particle-volume fraction (PVF) or $\phi_v$ and particle to fluid density ratio of $\phi_c = 10^{-3}$ and $C^+ = 2700$, respectively, were employed to allow the use of a two-way coupling approach between the particles and the carrier phase. It is shown that, regardless of the type of geometric perturbation prescribed, the injection of solid particles exhibited a strong attenuating effect of the turbulent intensity of the flow, as well as a turbulent skin friction drag reduction. These findings reinforce the concept of a selective stabilising effect induced by the solid particles. In this case, the PMF played an important role on the seemingly selective modulation of the turbulent activity.

1Universidad Nacional de Colombia under grant ”Asistente Docente” and COLCIENCIAS under grant ”Doctorados Nacionales call number 757”
Restriction 1 angles, length scales and frequencies within the given region of the TBL. This can potentially explain the modification in mean velocity profiles.

two-point correlations of the fluctuating velocities are used in elaborating on the dominate coherent structure configurations such as inclination angles, length scales and frequencies within the given region of the TBL. This can potentially explain the modification in mean velocity profiles.

This work was supported by NSF Grant 1604978

Analysis of Varying-Phase Opposition Control with Spatial Scale Restriction 1. SIMON TOEDTLI, BEVERLEY MCKEON, Caltech — This study considers a generalized version of the opposition control scheme (Choi et al, J Fluid Mech, 1994) from a Fourier domain perspective. Recent work (Toedtli et al, PRF, 2019) has shown that the effectiveness of the controller strongly depends on the relative phase between sensor measurement and actuator response, but an understanding of the underlying physics proves difficult so long as the controller simultaneously acts on a large number of spatial scales. We therefore consider here controllers with spatial scale restrictions and show that such controllers are capable of substantially altering the flow structure and drag. We first focus on the adverse scenario, where control leads to a pronounced drag increase, and use a combination of numerical simulation and modal analysis to shed light on the mechanisms underlying the change in drag. Insights obtained from the drag-increasing scenarios may help guiding the search for scale-restricted controller parameters that lead to drag reduction, which would be an important step towards a practical implementation of the control scheme.

This work is supported by the Air Force Office of Scientific Research through AFOSR grant number FA 9550-16-1-0361.

Control of a Turbulent Boundary Layer Separation Bubble by Shortfin Mako Shark Skin 1. AMY LANG, LEONARDO SANTOS, ANDREW BONACCI, JACOB PARSONS, University of Alabama — It has been demonstrated that real shortfin mako shark skin can control turbulent boundary layer separation due to the passive actuation of the scales in the presence of reversing flow. Unlike vortex generators, this passive flow-actuated mechanism functions locally at the point where there flow needs to be controlled and this study demonstrates that shark skin is capable of controlling the flow even downstream of the point where separation is already occurring. As in previous studies, shark skin specimens were mounted to a flat plate and placed in a tripped turbulent boundary subjected to an adverse pressure gradient induced by a rotating cylinder. DPIV experiments were conducted in a water tunnel facility for three different Reynolds numbers (on the order of 10^5) with different strengths of adverse pressure gradient to measure the control the presence of the shark skin had on the flow separation when the skin was placed on the downstream half of a quasi-steady turbulent separation bubble. Results confirm that the shark skin is able to control the flow by impeding the reversing flow near the surface. Furthermore, wall skin friction was calculated showing that the presence of the skin lowered the skin friction to the near zero vicinity but prevented it from going significantly negative as on the smooth wall cases.

Research funded by Army Research Office W911NF-15-1-0556

Effect of mass transfer on aeroheating in hypersonic chemically reacting boundary layers 1. MONA KARIMI, STC, NASA Ames Research Center, JOSEPH SCHULZ, AMA, NASA Ames Research Center, NAGI MANSOUR, NASA Ames Research Center — At atmospheric entry hypersonic speeds, ablation as well as surface catalycity will impact boundary layer aeroheating. Outgassing occurring from an ablative surface in planetary entry environment introduces a rich set of problems incorporating thermodynamic, fluid dynamic, and material pyrolysis. Although it is established that mass injection diminishes the wall heat transfer via convective blockage, understanding the underlying physical mechanism of the mass injection-induced boundary layer turbulence is still unresolved. To properly characterize the aerothermal environment and the required protection system, it is important to investigate gas-surface interaction models that inherently couple material response and boundary layer physics. The present study examines the aeroheating budget in hypersonic boundary layer with mass transfer produced material pyrolysis that reacts with the boundary layer environment. A coupled simulation of a chemically reacting viscous Navier-Stokes solver of the boundary layer over a pyrolyzing material is analyzed.

This research is supported by NASA’s Entry System Modeling (ESM) project under the Game Changing program.

ABSTRACT WITHDRAWN

Comparison of turbulent/non-turbulent interfaces in an adverse and zero pressure gradient turbulent boundary layer 1. JONGMIN YANG, JINYUL HWANG, MIN YOON, HYUNG JIN SUNG, KAIST, HYUNG JIN SUNG TEAM — The turbulent/non-turbulent interfaces of the zero pressure gradient (ZPG) and adverse pressure gradient (APG, \( \beta = 1.45 \)) turbulent boundary layers (TBLs) are explored using the direct numerical simulation data (Re76 = 830), where \( \beta \) is the Clauser pressure gradient parameter. The interfaces are extracted by the method based on the entrophy magnitude. Depending on the entrophy, the outer boundary layer flow can be classified into free stream, boundary layer wake, and intermittent flow regimes. In addition, we analyze the behavior of the intermittent flow regime by changing the threshold. The fractal dimension is obtained by using the box-counting algorithm. The fractal dimensions in the APG and ZPG TBLs are constant over the long range of the box size. The interfaces of the APG and ZPG TBLs show the monofractal behaviors. The geometric complexity of the interfaces in the APG and ZPG TBLs can be represented by the genus, which is defined by the number of handles in the geometric object. The genus in the APG TBL is larger than that in the ZPG TBL. The geometric complexity of the intermittent flow regime is increased in the APG TBL. In addition, we examine the projection area and the volume of the genus and the pockets to analyze the entrainment process in the APG and ZPG TBLs.

This study was supported by a grant from the National Research Foundation of Korea (No. 2019M3C1B7025091), and partially supported by the Supercomputing Center (KISTI).

ABSTRACT WITHDRAWN
3:44PM M01.00025 Influence of Splitter Plate Geometry on a Multi-Stream Jet Nozzle

EMMA GIST, Syracuse University, CORY STACK, The Ohio State University, DOMINIC DIDOMINIC, TYLER VARTABEDIAN, SETH KELLY, Syracuse University, DATTA GAITONDE, The Ohio State University, MARK GLAUSER, Syracuse University — Current and future aircraft designs are focusing on the integration of propulsion systems into the airframe to maximize efficiency and provide a stealth profile. This integration has led to different nozzle configurations such as the Multi-Aperture Rectangular Single Expansion Ramp Nozzle (MARS) where the nozzle has a primary and secondary bypass. This examination focuses on the reintroduction of the secondary subsonic bypass with the supersonic core flow and their merging behind a splitter plate. The two streams differ from each other in all aspects, including the velocity, pressure and density. Large-Eddy simulations (LES) have shown that splitter plate thickness affects the formation and evolution of large-scale structures in the presence of the shock and expansion wave system. This also impacts the acoustics as well as the mechanical loading due to the unsteady fluctuations in the flow. This study focuses on using thinner splitter plates and the introduction of passive control by a geometric change on the end of the splitter plate where the streams coalesce. This study aims to provide insight into the physics associated with the complex merging flows and how a splitter plate can be used to manipulate the flow to reduce noise without compromising performance.

1This study was provided funding via research grants by the Air Force Office of Scientific Research (AFOSR) grant number FA9550-19-1-0081 (Program Manager: Dr. Greg Abate) and Spectral Energies LLC.

3:45PM M01.00026 Computational Study of Film Cooling Performance of a Gas Turbine: An application of the Transverse Jets

GERMAN SIERRA-VARGAS, CARLOS DUQUE-DAZA, Dept. of Mechanical and Mechatronics Engineering, Universidad Nacional de Colombia — Film cooling technology based on transverse jets is employed to provide and control a protective film that remains attached to the surface and therefore to reduce the heat transfer from a high temperature main flow to the surface to be protected. Aiming to assess the impact of the jet-to-crossflow velocity ratio on the film cooling effectiveness, a number of numerical experiments were performed using four jet-to-crossflow velocity ratios over a NACA 4412 cascade vane. A computational model based on finite volume discretization, employing a WALE turbulence model, was prescribed to solve an incompressible flow on a 3D structured mesh. A passive scalar was included in the model to simulate temperature and transport of energy. Y+ values and Courant number were limited, in order to ensure convergence. Comparisons were made between the jet trajectory and the friction coefficient, evidencing how the mid-line of the cooling jet yields regions of boundary-layer separation and re-attachment. Analysis of the boundary-layer behavior indicates a relation with the local convective coefficient increments. Moreover, the results showed how the film cooling decreased the heat transfer at the region near the injection, but increased detrimentally the heat flux at the end of the vane.

3:46PM M01.00027 Multi-Jet Impingement Array Performance

ESCALLE THIBAUD, St. Cyr Military Academy, DAVID HELMER, MICHAEL BENSON, U.S. Military Academy — Impinging jets are frequently used in applications requiring cooling, and the design of such arrays requires understanding of both fluid dynamics and convective heat transfer. While impinging jet arrays have been extensively studied historically, there remain relatively few combined velocity and heat transfer datasets. This report presents such coupled measurements for an impinging jet array, including three-dimensional, three-component velocity measurements acquired using Magnetic Resonance Velocimetry, as well as full-field heat transfer measurements acquired with steady-state IR thermography with a joule-heating boundary condition. The goal of this measurement is to provide a benchmark dataset against which future experiments and especially simulations can be validated in detail.

1Combat Capabilities Development Center: Armaments

3:47PM M01.00028 Viscous elastic fluid jets induced by sudden acceleration

YOSHIYUKI TAGAWA, ANDRES FRANCO-GOMEZ, HAJIME ONUKI, YUICHIRO NAGATSU, Tokyo University of Agriculture and Technology — Modern interest for 3D-manufacturing applications requires controlled ejection of liquids with viscous non-Newtonian properties, such as polymer solution and molten resin. In this study, we compare jet evolutions of two viscous polymer solutions with different elasticity but similar shear-thinning properties (i.e. elastic and inelastic). Both jets are ejected by using our novel jet generation system employing an impulsive force (Onuki & Tagawa, Phys. Rev. Applied, 2018). The inelastic solution eventually pinch-off into droplets. In contrast, remarkably, jets of the elastic solution completely retract after ejection, even though the initial velocity of the jet is high (>10 m/s). We rationalize these behaviors by considering high elongational rate of liquids, which is beyond an explored range of existing studies. This contribution may open a new door for developing new additive-printing systems.

1JSPS KAKENHI Grant No. 17H01246

3:48PM M01.00029 Eddy viscosity for resolvent analysis of turbulent jets

ETHAN PICKERING, GEORGIOS RIGAS, Caltech, OLIVER SCHMIDT, University of California, San Diego, DENIS SIPP, ONERA — The French Aerospace Lab. TIM COLONIUS, Caltech — Resolvent modes of turbulent jets have shown striking qualitative agreement with data-deduced modes, found via spectral proper orthogonal decomposition (SPOD), of high-fidelity, large-eddy simulations (LES), however, quantitative comparisons are still lacking. The discrepancy is linked to the presence of spatially colored noise inherently contained within SPOD modes, found via spectral proper orthogonal decomposition (SPOD), of high-fidelity, large-eddy simulations (LES), however, quantitative comparisons are still lacking. The discrepancy is linked to the presence of spatially colored noise inherently contained within SPOD modes. The optimization is applied to Mach 0.4, 0.9, and 1.5 round, isothermal, turbulent jets, using five eddy-viscosity models: linear damping, a relation with the local convective coefficient increments. Moreover, the results showed how the film cooling decreased the heat transfer at the region near the injection, but increased detrimentally the heat flux at the end of the vane.

1Research support from the Office of Naval Research (Grant No. N00014-16-1-2445). E.P. supported by the Department of Defense, National Defense Science & Engineering Graduate Fellowship Program.
Between Quasi-Steady and Unsteady Dynamics

1 time-averaged flow fields are analyzed to understand the evolution of tip vortex under confinement. These contributions are measured in a water tunnel, at Reynolds numbers on the order of $10^4$, for regions spanning the suction surface of a delta wing with leading-edge sweep angle of 50 degrees. It is found that the balance depends on the chordwise position along the wing, with the shear layer contributing to LEV growth primarily near the apex of the wing, and transport further downstream dominated by flow three-dimensionality and interaction with the surface. Understanding these interactions provides a foundation for the design of flow control strategies and the prediction of aerodynamic loads and their fluctuations due to exogenous inputs.

1 Belin Blank Center: Secondary Student Training Program

3:50PM M01.00031 Fluid forces and flow transitions for a NACA0012 hydrofoil at low Reynolds numbers 1, Siddharth Gupta, Jisheng Zhao, Mark Thompson, Monash University, Australia, Atul Sharma, Amit Agrawal, IIT Bombay, India, Kerry Hourigan, Monash University, Australia — A study has been conducted to investigate the effect of angle of attack ($\alpha$) on the hydrodynamic performance and wake structure of a static NACA0012 hydrofoil in a free-stream flow at low Reynolds number ($Re$). The investigation employed water-channel experiments and in-house numerical simulations (based on an immersed interface method) over the angle of attack range of $0^\circ \leq \alpha \leq 90^\circ$ and the Reynolds number range of $2000 \leq Re \leq 10,000$. The angle of attack of a foil is an important parameter affecting the fluid dynamics and fluid-structure interaction; however, this problem has been poorly understood at low Reynolds numbers and particularly at large angles of attack, despite its importance in numerous applications, such as fish-like locomotion, autonomous underwater vehicles, bird-insect flights, micro-air vehicles, and wind turbines. The present findings showed that there exist different flow regimes and transitions over the $\alpha$ and $Re$ ranges investigated: e.g., a laminar flow regime is observed for $0^\circ \leq \alpha \leq 5^\circ$, followed by a transition regime prior to three distinctly different vortex shedding modes I, II and III for higher angles of attack. More details will be presented at the Division of Fluid Dynamics Meeting.

1 IITB-Monash Research Academy, INDIA

3:50PM M01.00032 Jacobi polynomial solution technique for the unsteady aerodynamics of porous airfoils 1, Rozhin Hajian, Harvard University, Peter J. Badoo, Imperial College London, Justin W. Jaworski, Lehigh University — Recent research has uncovered analytic solution forms for the flow field past a thin airfoil with an arbitrary porosity distribution. However, efforts to extend this work to unsteady flows or airfoil motions fail due to the presence of extra terms in the singular integral equation that are not readily treated analytically. To circumvent this issue, the bound vorticity along the chord is expanded as a series of weighted Jacobi polynomials. Analytic expressions for the parameters of the Jacobi polynomials are derived via asymptotic analysis. This approach is shown to be valid for static airfoils in steady flows with either continuous or discontinuous porosity distributions. A numerical validation is presented that demonstrates the spectral convergence of the scheme. The mathematical method is then extended to consider the unsteady motions of porous airfoils where the classical singular integral approach breaks down.

1This work was supported in part by the National Science Foundation under grant number 1805692.

3:52PM M01.00033 Confinement effects on the development of the tip vortex of an elliptical hydrofoil 1, Praveen Kumar, Krishnan Mahesh, University of Minnesota — Tip vortices are widely studied due to their relevance to many engineering applications. In many cases, e.g. ducted propulsors, tip vortices evolve under confinement. The effects of confinement on the development of tip vortex is the subject of the present work. Large eddy simulations are performed for flow over a confined elliptical hydrofoil at an incidence angle of 12 degrees and a Reynolds number of 0.9 million based on root chord length and freestream velocity. Two different cases of tip gap, i.e. the perpendicular distance between the hydrofoil tip and the bottom wall, are simulated and compared to the experiments of Boulon et al. (J. Fluid Mech. 1998, 390: 1-23), who studied confined effects on tip vortex cavitation. Instantaneous and time-averaged flow fields are analyzed to understand the evolution of tip vortex under confinement.

1This work is supported by the Office of Naval Research.

3:53PM M01.00034 ABSTRACT WITHDRAWN —

3:54PM M01.00035 ABSTRACT WITHDRAWN —

3:55PM M01.00036 Analysis of the Streamwise-Oscillating Cylinder Wake: Interplay Between Quasi-Steady and Unsteady Dynamics 1, Maysam Shamai, Caltech, Scott Dawson, Illinois Institute of Technology, Igor Mežić, University of California, Santa Barbara, Beverley Mckeeon, Caltech — The flow around a cylinder oscillating (surging) in the streamwise direction with a frequency, $f_s$, much lower than the shedding frequency, $f_s$, has been relatively less studied than the case when these frequencies have the same order of magnitude. We combine particle image velocimetry and Koopman Mode Decomposition to investigate the cylinder wake for nominal parameters $f_s/f_s \approx 0.04 - 0.2$ and mean Reynolds number, $Re \approx 900$. The amplitude of oscillation is such that the instantaneous Reynolds number is far from the critical value. Characterization of the wake reveals a range of phenomena associated with nonlinear interaction of the two frequencies, including amplitude and frequency modulation. We perform analyses in multiple frames of reference to motivate use of the cylinder-fixed frame. Utilizing this frame, we present a scaling parameter and associated transformation in order to relate the unsteady, or forced, dynamics to that of a quasi-steady, or unforced, system. Implications for Koopman analysis of the flow around a moving body will be discussed. This work is supported under ARO grant W911NF-17-1-0306.

1This work is supported under ARO grant W911NF-17-1-0306.
1AMIR CHIZFAHM, RAJEEV JAIMAN, University of British Columbia — Fluid-structure interaction of an elastically-mounted sphere exhibits a wide range of complex flow-induced vibration (FIV) regimes. Unlike a vast amount of literature available on the vortex-induced vibration of an elastically-mounted circular cylinder, such studies on a sphere are limited. We aim to understand the fundamentals of new vortex-shedding modes and coupled dynamics pertaining to the FIV response of a freely vibrating sphere in all three spatial directions, using a body-fitted finite-element based fluid-structure interaction framework. To predict and analyze the vortex synchronization regimes and the wake patterns, the FIV response of the sphere at a low mass ratio is investigated over a broad range of reduced velocity and Reynolds number. We find that the sphere begins to move along a linear trajectory with hairpin vortex-shedding mode, finally transforming into a circular trajectory with spiral mode in its stationary state. We systematically examine these mode transitions and motion trajectories in the three degrees-of-freedom for the Reynolds number up to 30,000 which has not been studied in detail in the literature.

1RAJEEV JAIMAN, University of British Columbia — Fluid-structure interaction of an elastically-mounted sphere exhibits a wide range of complex flow-induced vibration (FIV) regimes. Unlike a vast amount of literature available on the vortex-induced vibration of an elastically-mounted circular cylinder, such studies on a sphere are limited. We aim to understand the fundamentals of new vortex-shedding modes and coupled dynamics pertaining to the FIV response of a freely vibrating sphere in all three spatial directions, using a body-fitted finite-element based fluid-structure interaction framework. To predict and analyze the vortex synchronization regimes and the wake patterns, the FIV response of the sphere at a low mass ratio is investigated over a broad range of reduced velocity and Reynolds number. We find that the sphere begins to move along a linear trajectory with hairpin vortex-shedding mode, finally transforming into a circular trajectory with spiral mode in its stationary state. We systematically examine these mode transitions and motion trajectories in the three degrees-of-freedom for the Reynolds number up to 30,000 which has not been studied in detail in the literature.

1Membership Pending

3:57PM M01.00039 What is a “Length Scale” in Variable Density Turbulence?1. DONGXIAO ZHAO, HUSSEIN ALUIE, University of Rochester — A “length scale” in a fluid flow does not exist as an independent entity but is associated with the specific flow variable being analyzed. While this might seem obvious, we often discuss the “inertial range” or the “viscous range” of length scales in turbulence as if they exist independently of a flow variable, which in incompressible turbulence is the velocity field. How should we analyze “length scales” in flows with significant density variations, such as across a shock or in multiphase flows? A choice can be made according to the so-called inviscid criterion. It is a kinematic requirement that a scale decomposition yield negligible viscous effects at sufficiently large “length scales.” Recently, we proved that a Hesselberg-Favre decomposition satisfies the inviscid criterion, which is necessary to unravel inertial-range dynamics and the cascade. We present numerical demonstrations of those results, where we also show that other commonly used decompositions can violate the inviscid criterion and, therefore, are not suitable to study inertial-range dynamics in variable-density turbulence.

1Supported by DOE grant DE-SC0014318

3:59PM M01.00040 Homogeneous variable-density turbulence with asymmetric initial density distributions1. DENIS ASLANGIL, Lehigh University, DANIEL LIVESCU, Los Alamos National Laboratory, ARINDAM BANERJEE, Lehigh University — In most natural and engineering applications, turbulent mixing occurs between unbalanced amounts of two or more miscible fluids of different densities. For example, during Rayleigh-Taylor and Richtmyer-Meshkov instabilities, the mole fraction percentages of the pure fluids change from zero to unity from edge to edge within the mixing layer. In this study, we investigate the effects of differential amounts of mixing fluids on the evolution of HVDT by using high-resolution direct numerical simulations (up to 20483) for two different density ratios- 1.1 and 7.1. Three cases with different initial compositions characterized by an initial composition ratio ($\chi$ = mole fraction of the heavy fluid/ mole fraction of heavy fluid) was chosen for each density ratio; a heavy fluid dominated case (HF) with $\chi$ = 3, a light fluid dominated case (LF) with $\chi$ = 1/3 and the classical HVDT case where $\chi$ = 1. It is found that at large density ratios, upon increasing the initial amount of the pure light fluid, the turbulence kinetic energy generation is enhanced, whereas upon increasing the initial amount of the pure heavy fluid, the turbulence generation is suppressed. In addition, it takes longer for turbulence to disperse into the regions of heavy fluid compared to regions of light fluid.

1Authors acknowledge support from DOE/NNSA Grant # DE-NA0003195 and the NSF-CAREER Award #1453056. DL is an employee of Triad National Security, LLC which operates LANL under Contract #89233218CNA000001 with the U.S.DOE NNSA.

4:00PM M01.00041 A variational level set method without reinitialization for predicting equilibrium interfaces over arbitrary textured surfaces1. KARIM ALAME, SREEVATSA ANANTHARAMU, KRISHNAN MAHESH, University of Minnesota — A robust numerical methodology to predict equilibrium interfaces over arbitrary solid surfaces is developed. The kernel of the proposed method is the distance regularized level set equations (DRLSE) with techniques to incorporate the no-penetration and mass-conservation constraints. In this framework, we avoid reinitialization typically used in traditional level set evolution algorithms. The method is second-order accurate and requires only central difference schemes. The application of the method, in the context of Gibbs free energy minimization, to obtain liquid-air interfaces is validated against existing analytical solutions. The capability of our current methodology to predict equilibrium shapes over both structured and realistic rough surfaces is demonstrated.

1This work is supported by the Office of Naval Research (ONR).
Investigation of Mixing Law Efficacy for Hydrodynamic Simulations and Associated Compressibility Implications. CALEB WHITE, HUMBERTO SILVA III, Sandia National Laboratories, PETER VOROBIEFF, Department of Mechanical Engineering, University of New Mexico — A computational simulation of various mixing laws for gaseous equations of state (EOS) using planar traveling shocks for multiple mixtures in three dimensions (3D) is analyzed against nominal experimental data. Numerical simulations utilize the Sandia National Laboratories (SNL) shock hydrodynamic code CTH and other codes including the SNL thermochemical equilibrium code TIGER and the uncertainty qualification (UQ) and sensitivity analysis code DAKOTA. The mixtures are: a 1:1 and a 1:4 molar mixture of helium (He) and sulfur hexafluoride (SF\textsubscript{6}). The mixing laws to be analyzed are the ideal gas law, Amagat's Law, and Dalton's Law. Examination of the experimental data with TIGER revealed that the shock strength should not be considered when the mixture is non-ideal as the compressibility factor, \( \zeta \), was essentially unity (\( \zeta \approx 1.02 \)). Strikingly however, experimental results show that neither Dalton's nor Amagat's Law are able to accurately predict the properties of the shocked mixture. A methodology is being developed to possibly optimize the various mixing laws with computational results for the data set investigated. Lastly, a framework for future sensitivity and uncertainty quantification analysis will be established.

Effect of counter-gradient subgrid-scale transport on turbulent mixing. SIDHARTH GS, RAYMOND RISTORCELLI, Los Alamos National Laboratory — The present work explores the effect of subgrid-scale models on the statistics of turbulent mixing of passive and active scalars. We compare the commonly employed gradient diffusion model against the non-linear gradient model (related to Clark model/Finite-scale equations). The aim is to investigate the consequence of isotropic eddy viscosity/scalar diffusivity versus a tensorial viscosity and diffusivity that permits counter-gradient transport of resolved-scale variables. For an isotropic turbulent flow, an gradient diffusion model, the non-linear gradient model can be shown to preserve the combined supergrid and subgrid scalar variance to the leading order in filter width. Therefore, the effect of the two classes of models on the evolution of the scalar variance (passive and active) is contrasted. Furthermore, in the active scalar case (variable-density mixing), we compare the turbulence and mixing statistics in the Reynolds- versus Favre-filtered representation of large-scale velocity and scalar variables (Sidharth GS and Candler JFM (2018)). The role of variable-density subgrid acceleration on the dynamics of subgrid velocity variance is of particular interest and compared with the well-studied specific-stress based production term.

Topological classification of recurrences in turbulent flows. NAZMI BURAK BUDANUR, GÖKHAN YALNIZ, BJÖRN HOF, IST Austria — In recent years, numerical discoveries of unstable time-periodic solutions in various shear flow simulations have sparked hopes of developing a chaos theoretic understanding of turbulence. For many cases of interest, however, the standard tools of chaos theory, such as Poincaré sections, were insufficient for uncovering a complete picture of turbulent dynamics due to its high dimensionality. As a result, the discoveries of periodic orbits in turbulent flows have remained at an illustrative level with no obvious paths toward their utilization in turbulence modeling and control. One simple question one might ask is whether the turbulent dynamics transiently approximate periodic solutions, and if so, how frequently? We argue that a systematic study of this problem requires a method for unsupervised identification of geometric similarities between periodic orbits and turbulent trajectory segments in the system's state space. We will demonstrate with examples that topological data analysis methods can be employed for this purpose.

Probe into the gas leakage dynamics from the bubbly wake of a ventilated supercavity. SIYAO SHAO, JIARONG HONG, University of Minnesota — Understanding the liquid-gas interface instability and associated gas leakage mechanisms is critical for developing new strategies for sustainable ventilated supercavitation in practical applications. However, despite recent effort from Wu et al [JFM, 2019, 862, 1135-1165], to directly characterize the gas leakage through the cavity internal flow measurement is challenging, particularly across a broad range of cavity regimes. Here we probe into the gas leakage mechanism by investigating bubbly wake generated from a ventilated supercavity with various closure modes including re-entrant jet, twin and quad vortex closures. The size and shape of bubbles and their 3D distribution in the wake are captured using a high speed digital inline holography (DIH). The instantaneous gas leakage rate, estimated from the size and velocity of bubbles at each time instant, shows a strong intermittent behavior while the average gas leakage from DIH agrees well with the ventilation input under all experimental conditions. In addition, the detailed spatial and temporal characteristics of bubble distribution in the wake are found to vary under different closure conditions, connecting strongly with the interface instability and bubble breakup mechanism at the closure of the supercavity.

Kelvin wave generation on vortices in Bose-Einstein condensates. SCOTT STRONG, Applied Mathematics and Statistics, Colorado School of Mines, LINCOLN CARR, Department of Physics, Colorado School of Mines — A single line of concentrated vorticity is an open and fundamental problem in the study of superfluid turbulence. The local induction approximation, or LIA, is a straightforward integrable model of curvature induced flow. Here, the curvature and torsion evolve under a cubic focusing nonlinear Schrödinger equation whose wealth of conservation laws are thought to artificially constrain interactions between helical modes. Our work describes LIA as the lowest-order approximation in a fully nonlinear expansion of curvature induced motion honoring archlength conservation present in the Hamiltonian formulation of inviscid fluid dynamics. These higher-order corrections are accurate at scales where LIA is not, and accounts for locally induced flows and contributions due to the vortex core. Our fully nonlinear model predicts that traveling waves of localized curvature seek to transport bending along the vortex. Simulations show dynamics similar to those seen post-reconnection in vortices generated by obstacles and cavitation in classical flows. In ultratqumon turbulent tangles, energy transfer between helical Kelvin modes of vortex lines permits free decay and our relaxation of bending via Kelvin wave generation may be its most primitive manifestation.
4:08PM M01.00049 A formation time scale for vortex rings generated by pulsed planar jets. BEN STEINFURTH, TIM GEFFKE, JULIEN WEISS, Technische Universität Berlin — The flow field of a pulsed planar jet emitted from an outlet of high aspect ratio is studied experimentally. Considering that the effectiveness of various flow control applications is determined by large-scale coherent vortex structures, the objective of this study is to shed some light upon the generation mechanisms of these vortices. First, flow visualizations are conducted with pulsed jets issued into a steady water tank, verifying that the concept of an optimal generation time scale ensuring the exclusive generation of a leading vortex ring exists. Then, quantitative measurements are performed employing particle image velocimetry. Based on the derivation of flow diagnostics and additional extensive hotwire measurements, the following main conclusion can be drawn: increasing the pulse width, i.e., the amount of ejected fluid of a pulsed planar jet results in saturation of the leading vortex ring in terms of its dimensions and entrainment characteristics. Beyond a certain pulse width, a trailing structure with properties similar to a steady planar jet occurs. The shear rates of this trailing jet are well below those observed inside the vortex ring. Thus, jet modulation with the identified optimal pulse width may be beneficial from a flow control perspective.

4:09PM M01.00050 Semi-Lagrangian Lattice Boltzmann Method for Compressible Flows. DOMINIK WILDE, University of Siegen, ANDREAS KRAEMER, National Heart, Blood and Lung Institute, National Institutes of Health, HOLGER FOYSI, University of Siegen — The lattice Boltzmann method (LBM) is an established tool for the simulation of weakly compressible flows. However, in the field of compressible flows the LBM is still lacking a widely accepted framework, which is why it is an active field of research. On the one hand, traditional LBM solvers with an exact propagation of the distribution function values usually require large velocity sets. On the other hand, Eulerian solvers like finite volume or finite difference LBMs suffer from high computation costs. We propose a semi-Lagrangian streaming step allowing for unstructured grids and on non-integer-based velocity sets. This procedure effects small numerical dissipation, while the spatial order of convergence can be increased by the use of high-order interpolation polynomials in combination with an appropriate choice of support points. The semi-Lagrangian LBM circumvents the costly application of explicit time integration in Eulerian schemes. Instead, from an algorithmic point of view, the semi-Lagrangian LBM is still close to the original LBM formulation. Simulations of a Sod shock tube, a 2D Riemann problem, a shock-vortex interaction, and a 2D airfoil confirm the newly introduced semi-Lagrangian LBM to be appropriate for the calculation of compressible flows.

4:10PM M01.00051 ABSTRACT WITHDRAWN —

4:11PM M01.00052 Princeton High Reynolds Number Supertank facility. Konstantinos Steiros, Marcus Hultmark, Princeton University, Mechanical and Aerospace Engineering — Characterizing the aerodynamic behavior of large structures presents a major challenge in many engineering fields, including modern wind engineering, architecture, and urban planning research. The combination of enormous Reynolds numbers, relatively small Mach numbers, and unsteady or periodic events, render such flow measurements practically impossible in a conventional test facility. To address this issue, we present a novel recirculating wind tunnel facility, the Supertank, where the gauge pressure will be varied from 0 to 80 bar, enabling the testing of a large range of Reynolds numbers. The novelty of this facility lies in its large size, allowing a test section of 0.88 X 0.88 m^2 cross section and 7.5 m length, and in its easily accessible design. In that manner, series of wind turbines and even small wind farms will be able to be characterized at high Reynolds numbers, and it will serve as an ideal facility to evaluate numerical models and simulations. Several technical details of the facility will be discussed, along with the potential that this facility will unlock for applied and fundamental research.

4:12PM M01.00053 Leverage the Capability of Princeton Superpipe. LiuYang Ding, Alexander Pique, Simere Genet, Daniel Hoffman, Marcus Hultmark, Alexander Smits, Princeton University, Princeton University TEAM — The Superpipe facility at Princeton utilizes compressed air as the working fluid to obtain high-Reynolds-number turbulence. It comprises a recirculating pressure vessel that can hold up to 220 atm, and a test pipe of 200 diameter long inside the pressure vessel. The range of bulk Reynolds number achievable is 8x10^7 to 6x10^9. Previous measurements in the Superpipe have only employed hot wires and only investigated equilibrium flows. We now report our progress in maximizing the capability of Superpipe towards optical measurement of non-equilibrium turbulence at high Reynolds numbers. We designed a new traversing system with a miniature rail mounted inside the test pipe, which allows test models to travel over a large axial distance. The blockage ratio of the rail is 0.6% of the pipe cross section. A linear driving stage is placed in the diffuser section and is 16 diameters downstream of where the flow is sampled. We will present preliminary hot-wire data of turbulence past a streamlined body of revolution, and compare it with PIV measurement of the same flow in a water pipe. In addition, the design of a new PIV system for the Superpipe will be presented. Details regarding imaging and illumination with optical fibers, calibration procedure, and seeding method will be discussed.

1 ONR N00014-17-1-2309

Monday, November 25, 2019 3:20PM - 4:16PM — Session M02 Flash Oral Presentations: Geophysical Flows

3:20PM M02.00001 Passive vortical flows enhance mass transfer in a coral colony. MD MONIR HOSSAIN, ANNE STAPLES, Virginia Tech — Corals are sessile and rely on the surrounding ocean flow to obtain nutrients and carry out their other physiological functions. Recent studies have shown that corals in low flow conditions can stir the water column, creating vortical flows that enhance mass transfer rates by up to 400% (Shapiro et al., PNAS, 2014). Here, we perform three-dimensional immersed-boundary simulations of the flow through a single Pocillopora meandrina colony under high flow conditions. We demonstrate that the passive geometric features of the branching colony produce highly vortical internal flows. This enhances mass transfer at the interior of the colony and compensates almost exactly for flow speed reductions there of up to 64%, resulting in the advection time scale remaining roughly constant throughout the colony. We further compute the transport of a passive scalar from the surface of the colony under idealized sinusoidal oncoming flow conditions and find a double-peak concentration profile in the interior of the colony.

4:10PM M01.00051 ABSTRACT WITHDRAWN —

4:11PM M01.00052 Princeton High Reynolds Number Supertank facility. Konstantinos Steiros, Marcus Hultmark, Princeton University, Mechanical and Aerospace Engineering — Characterizing the aerodynamic behavior of large structures presents a major challenge in many engineering fields, including modern wind engineering, architecture, and urban planning research. The combination of enormous Reynolds numbers, relatively small Mach numbers, and unsteady or periodic events, render such flow measurements practically impossible in a conventional test facility. To address this issue, we present a novel recirculating wind tunnel facility, the Supertank, where the gauge pressure will be varied from 0 to 80 bar, enabling the testing of a large range of Reynolds numbers. The novelty of this facility lies in its large size, allowing a test section of 0.88 X 0.88 m^2 cross section and 7.5 m length, and in its easily accessible design. In that manner, series of wind turbines and even small wind farms will be able to be characterized at high Reynolds numbers, and it will serve as an ideal facility to evaluate numerical models and simulations. Several technical details of the facility will be discussed, along with the potential that this facility will unlock for applied and fundamental research.

4:12PM M01.00053 Leverage the Capability of Princeton Superpipe. LiuYang Ding, Alexander Pique, Simere Genet, Daniel Hoffman, Marcus Hultmark, Alexander Smits, Princeton University, PRINCETON UNIVERSITY TEAM — The Superpipe facility at Princeton utilizes compressed air as the working fluid to obtain high-Reynolds-number turbulence. It comprises a recirculating pressure vessel that can hold up to 220 atm, and a test pipe of 200 diameter long inside the pressure vessel. The range of bulk Reynolds number achievable is 8x10^7 to 6x10^9. Previous measurements in the Superpipe have only employed hot wires and only investigated equilibrium flows. We now report our progress in maximizing the capability of Superpipe towards optical measurement of non-equilibrium turbulence at high Reynolds numbers. We designed a new traversing system with a miniature rail mounted inside the test pipe, which allows test models to travel over a large axial distance. The blockage ratio of the rail is 0.6% of the pipe cross section. A linear driving stage is placed in the diffuser section and is 16 diameters downstream of where the flow is sampled. We will present preliminary hot-wire data of turbulence past a streamlined body of revolution, and compare it with PIV measurement of the same flow in a water pipe. In addition, the design of a new PIV system for the Superpipe will be presented. Details regarding imaging and illumination with optical fibers, calibration procedure, and seeding method will be discussed.

1 ONR N00014-17-1-2309

Monday, November 25, 2019 3:20PM - 4:16PM — Session M02 Flash Oral Presentations: Geophysical Flows

3:20PM M02.00001 Passive vortical flows enhance mass transfer in a coral colony. MD MONIR HOSSAIN, ANNE STAPLES, Virginia Tech — Corals are sessile and rely on the surrounding ocean flow to obtain nutrients and carry out their other physiological functions. Recent studies have shown that corals in low flow conditions can stir the water column, creating vortical flows that enhance mass transfer rates by up to 400% (Shapiro et al., PNAS, 2014). Here, we perform three-dimensional immersed-boundary simulations of the flow through a single Pocillopora meandrina colony under high flow conditions. We demonstrate that the passive geometric features of the branching colony produce highly vortical internal flows. This enhances mass transfer at the interior of the colony and compensates almost exactly for flow speed reductions there of up to 64%, resulting in the advection time scale remaining roughly constant throughout the colony. We further compute the transport of a passive scalar from the surface of the colony under idealized sinusoidal oncoming flow conditions and find a double-peak concentration profile in the interior of the colony.
at an angle of $\pm \sqrt{\frac{1}{2}} \approx \pm 35.35^\circ$ to the direction of travel of the carrier wave. Moreover, we analyse in detail the single crossed-wave component and find that group dispersion decreases to a minimum at the nondispersive crossing angle of approximately $\pm 35.26^\circ$. Our results may motivate investigations in other physical media, governed by weakly nonlinear evolution equations and improve understanding of extreme event lifetime.

1UK EPSRC Studentship, Royal Academy of Engineering Research Fellowship, and the Italian Ministry of Education

3:25PM M02.00006 The Cooling Box Problem: A Freezing Lake in the Lab$^1$ JASON OLSTHOORN, KYLE GERRARD, EDMUND W. TEDFORD, GREGORY A. LAWRENCE, University of British Columbia — Recent field measurements have demonstrated that inland waters are warming rapidly and that seasonally ice-covered lakes are warming faster than those that do not. With limited field studies of lakes in harsh winter conditions, the impact of this changing environment is not clear, particularly from an ecological perspective. Similarly, the physical processes responsible for the warming within these cold-water bodies have not yet been identified. Our research focuses on the surface cooling processes occurring prior-to and immediately after ice formation, with a particular interest in the transport of heat. We model a cooling box problem, similar to the Rayleigh-Bernard problem. However, the nonlinear equation of state of freshwater complicates the traditional Rayleigh-Bernard analysis. We demonstrate that the nonlinear equation of state fundamentally alters the classical results of the Rayleigh-Bernard problem. Using a combination of linear stability analysis, numerical simulation and laboratory experiments, we quantify the mixing within the water column and the resultant surface heat transfer. Modified scaling laws for heat transport and energetics agree well with our data.

$^1$Natural Sciences and Engineering Research Council of Canada

3:26PM M02.00007 ABSTRACT WITHDRAWN
3:27PM M02.00008 Local flow and turbulence at a tidal energy conversion installation near a pier of an estuarine bridge¹, MARTIN WOSNIK, KAELIN CHANCEY, University of New Hampshire — Estuarine bridges could serve as ideal locations to deploy marine hydrokinetic (MHK) energy conversion systems. The hydrokinetic resource is typically strong at these narrow locations, the bridge piers can serve as supporting structure for both the bridge and turbines, and synergies exist in the permitting processes. The Living Bridge Project installed a hydrokinetic turbine on a floating platform at Memorial Bridge in Portsmouth, NH. The location is well-suit-ed as a tidal energy test site, reaching tidal current speeds greater than 2.5 m/s during spring ebb tides. In tidal estuaries the currents can vary significantly in space and time. Measurements were conducted with two acoustic Doppler current profilers (ADCP), mounted on the bow and stern of the platform, and with two acoustic Doppler velocimeters, mounted in various locations. The ADCPs indicated higher maximum current velocities and mean kinetic power density than prior nearby resource assessments. The ADV measurements yielded turbulence time and length scales consistent with estuary scales, e.g., width of the river and distances between the bridge piers. The tidal flow turbulence characteristics, such as the size and occurrence of coherent structures, affects the loading on the tidal turbine.

¹The project received funding from NSF-PFI (grant IIP 1430260), FHWA, NHDOT and DOE.

3:28PM M02.00009 Duffing equation describes sloshing experiments, KERSTIN AVILA, BASTIAN BAUERLEIN, Center of Applied Space Technology and Microgravity (ZARM), University of Bremen, 28359 Bremen, Germany — In nature and in engineering, periodic forcing leads often to nonlinear resonances which are challenging to model and predict. The Duffing equation \( (\ddot{x} + \beta \dot{x} + \alpha x + \omega^2 x = F \cos \omega t) \) is the simplest and most widely studied model describing such phenomena and is thus used in introductory courses to nonlinear dynamics. Despite numerous analytical and numerical studies across disciplines investigating its properties and dynamics, laboratory experiments exhibiting its dynamics are very scarce and, to our knowledge, all of them have been constructed to behave as a Duffing oscillator. We show that sloshing of water in a rectangular container driven by harmonic horizontal excitation is accurately described by the Duffing equation. The choice of the setup combined with automated flow visualizations and PIV-measurements allow it to reproduce key features of the Duffing oscillator experimentally. Besides the characteristic resonance curve we observe a hysteresis which increases with the driving amplitude and features transitions at exactly 90°-phase-lag, as well as period-three-motion. The experiments reach from linear and nonlinear dynamics fully described by the Duffing oscillator, up to highly nonlinear effects with e.g. breaking surface waves.

3:29PM M02.00010 Effect of spring non-linearity on vortex-induced vibration of a circular cylinder¹, RAHUL MISHRA, Monash University, Australia, RAJNEESH BHARADWAJ, IIT Bombay, India, MARK THOMPSON, Monash University, Australia — The vortex-induced vibration (VIV) of a circular cylinder subject to nonlinear structural support has been studied computationally for fixed mass ratio \((m^* = 2.546)\) in the two-dimensional Reynolds number regime. Unlike for the classic case for which the structure support consists of a spring and damper in parallel, this study considers a system composed of two springs and one damper, where the two springs are in parallel and the damper is in series with one of the springs. The arrangement of the springs and damper is similar to the Standard Linear Solid (SLS) model used for modelling the behavior of a viscoelastic material. The spring in series with the damper is linear and that parallel to the damper provides a non-linear force. The non-linear structural system is governed by the following three parameters: (a) the ratio of the spring constant \((R)\), (b) damping ratio \((\zeta)\), and (c) non-linearity strength \((\lambda)\). The focus of the present study is to examine the response of the cylinder to VIV subject to changing \(\zeta\) and \(\lambda\). The main feature of this non-linear system is its ability to sustain vibration for a greater range of flow velocity; potentially useful for vibratory energy extraction.

¹IITB-Monash Research Academy, India

3:30PM M02.00011 The flow surrounding fire whirls¹, ADAM D WEISS, PRABAKARAN RAJAMANICKAM, WILFRIED COENEN, ANTONIO L SANCHEZ, FORMAN A WILLIAMS, University of California San Diego — Despite significant research efforts, our current understanding of the flow structure and dynamics of fire whirls, including the reasons for their dramatic flame-lengthening effect and increased burning rate, is far from complete. The present study contributes to the needed understanding by investigating the steady axisymmetric structure of the cold outer flow surrounding fire whirls developing over localized fuel sources lying on a horizontal surface. Consideration is given to the three distinct flow regions arising from the disparity of length scales present in the problem, namely, an outer rotational inviscid region driven by the buoyant turbulent plume of hot combustion products that develops above the fire, a near-wall boundary layer, and a near-origin non-slender region where the radial inflow of the boundary layer collides. The resulting description, and in particular the terminal boundary-layer structure as the axis is approached, may be useful in future numerical computations of fire whirls.

¹This work was funded by the National Science Foundation through award #1916979 (Swirling dynamics in liquid-pool fires)

3:31PM M02.00012 Potential Fluid Mechanisms for Low Frequency Sound from Tornadoes¹, BRIAN ELBING, CHRISTOPHER PETRIN, Oklahoma State University, MATTHEW VAN DEN BROEKE, University of Nebraska-Lincoln — Tornado-producing storms have been observed to emit infrasound (sound at frequencies below human hearing) up to 2 hours before tornadoogenesis. Weak atmospheric attenuation at these frequencies allows for long-range detection. Hence, passive infrasonic monitoring could be a method for long-range studying of tornadoogenesis as well as tornado characterization. Identifying the fluid mechanism(s) that produce the infrasound is critical to enable such capabilities. Besides the characteristic resonance curve we observe a hysteresis which increases with the driving amplitude and features transitions at exactly 90°-phase-lag, as well as period-three-motion. The experiments reach from linear and nonlinear dynamics fully described by the Duffing oscillator, up to highly nonlinear effects with e.g. breaking surface waves.

¹NSF Grant 1539070 and NOAA NA18OAR4590307

3:32PM M02.00013 Uncertainty Quantification of Models for Ocean Surface Boundary Layer Turbulence, GREGORY WAGNER, RAFFAELLE FERRARI, ANDRE SOUZA, Massachusetts Institute of Technology — The atmosphere and ocean communicate through the ocean’s turbulent surface boundary layer (OSBL), and accurate models of OSBL turbulence are necessary for accurate climate prediction. In this talk we use a suite of Large Eddy Simulations of OSBL turbulence in a range of physical scenarios to optimize and estimate the uncertainty of free parameters in models for OSBL turbulent mixing designed to be embedded in ocean general circulation models. We evaluate deficiencies in the structure of several different OSBL turbulence models by comparing the dependence of optimal parameters on the targeted physical scenario. Our ultimate goal is to choose a best OSBL turbulence closure for implementation in a new Earth System Model being developed as part of the Climate Machine (CiMa) project.
3:33PM M02.00014 Coupled-mode flutter in wind turbine blades – numerical prediction and experimental evidence.1 YAHYA MODARRES-SADEGHI, TODD CURRIER, PIETER BOERSMA, BRIDGET BENNER, University of Massachusetts Amherst, XAVIER AMANDOLESE, Ecole Polytechnique, France — We present a model to predict the onset of couple-coupled flutter in wind turbine blades as well as the post-instability behavior of the blade. While linear models to predict the onset of wind turbine blade instabilities have been around for a while, nonlinear models are lacking. Besides, experimental results to show these instabilities and to validate the models for wind turbine blades do not exist. We discuss a nonlinear model to predict these instabilities and present experimental results to validate the model for blades of around 200 cm in length. These results and other tests with blades of around 200 cm in length. In both series of experiments dynamic instabilities are observed. The larger-scale tests enable us to observe the coupling of two blade modes in a coupled-mode response, as well as oscillations purely in the torsional direction at higher wind speeds. The coupled-mode flutter is also predicted in the nonlinear numerical model.

3:35PM M02.00015 Effects of Land Cover on Wind Profiles: Case Study at Kirkwood Iowa. ROBERT AHLMAN, WEI ZHANG, Cleveland State University, COREY D. MARKFORT, University of Iowa — The atmospheric boundary layer serves as the incoming flow and kinetic energy source for wind turbines and remains a particularly challenging flow to study in fluid dynamics research. The difficulty lies in not only the wide range of spatial and temporal turbulent scales that need to be resolved but also the effects of the underlying surface. In particular, it is not well understood how wind velocity profiles respond to complex terrain and variations in atmospheric thermal stability. This research aims to characterize wind profiles in the surface layer and assess the accuracy of commonly used metrics for various terrain and thermal stability conditions. Data sets were recorded by various instruments mounted on a 106-meter tall meteorological tower at the Kirkwood Community College in Cedar Rapids, Iowa. Vertical profiles of wind speed and temperature, filtered by the direction of the incoming wind, for an entire year have been analyzed. Standard metrics and well-established formulations were assessed for their ability to accurately describe the wind profiles for a variety of different conditions. This work helps to provide insights into the effects of complex terrain and the atmospheric thermal stability on wind profiles, crucial to onshore wind resource assessment.

3:35PM M02.00016 Effect of the Instabilities in the Overlying Atmospheric Boundary Layer on the Street Canyon Ventilation TADEU MENDONCA FAGUNDES, NEDA YAGHOOBIAN, JUAN ORDONÉZ, Florida State University — Human health and air quality in urban areas are important problems that are linked to the pollution dispersion and ventilation capacity of urban streets. The ventilation mechanism and transfer process within urban streets is in strong connection with the instabilities in the overlying atmospheric boundary layer. In this study, we aim to use computational modelling to investigate the coupling between the flow within the street canyons and the turbulent structures in the above canopy flow. The state of the atmospheric boundary layer over an idealized urban area is controlled by the condition of the upstream topography. The transport phenomena in urban canyons is examined under different upstream conditions to reveal the ventilation mechanism in urban areas.

3:36PM M02.00017 Wake recovery in collocated wind plants RAL BAYON CAL, HAWWA KADUM, Portland State University, MIKE QUIGLEY, GERARD CÓRTINA, MARC CALAF, University of Utah — Large eddy simulations approach is used to investigate the power production enhancement mechanisms in collocated wind plants in which twelve clusters of vertical axis wind turbines are collocated with a 3x4 horizontal axis wind turbine array. Three cases are studied: 1.) a standard wind plant (SWP), 2.) an aligned collocated wind plant (CW1|P1), and 3.) a staggered collocated wind plant (CW1|P2). A control volume analysis is employed to examine the energy balance and relevant terms for the various characteristic compounded wakes. The results show that collocated configurations have an averaged 48.5% higher power than the standard wind plant due to the faster wake recovery and improved vertical transport of mean kinetic energy. The collocated plants spatial heterogeneity is found to play a significant role in mean kinetic energy vertical transport advancement by increasing the dispersive stress with an average of 37.5% increase in the vertical kinematic shear stress from the standard wind plant, consequently reinforcing the mean kinetic energy flux which is the lead term in mean kinetic energy budget. This arrangement resulted in 4% higher power production for the aligned configuration than the staggered even though the latter has faster wake recovery.

3:37PM M02.00018 ABSTRACT WITHDRAWN

3:38PM M02.00019 On the near-wake characteristics of a tidal current model in a sheared turbulent inflow1 ASHWIN VINOD, ARINDAM BANERJEE, Lehigh University — Tidal current turbines deployed in tidal flows can be anticipated to encounter sheared and turbulent flow environments. Therefore, a thorough understanding of the implications of such operating conditions would be valuable in optimizing the performance and operational life of the installed turbines. The ongoing experimental work at Lehigh University aims to improve the understanding of tidal turbine performance, and the mechanism of momentum transfer in its near-wake in a controlled, sheared, turbulent inflow. A 1:20 laboratory-scale tidal turbine model with a rotor diameter of 0.28m is used in the experiments. An active grid type turbulence generator consisting of a series of five stepper motor-controlled horizontal winglet shafts is employed to generate a vertically sheared, turbulent inflow. To better control the shear profile, winglets with different sizes/solidities are utilized in the active grid. All flow measurements were carried out using an acoustic doppler velocimeter. In addition to performance metrics, and mean, turbulent wake characteristics, contributions of the different terms in mean momentum and kinetic energy equations are also examined to better capture the process of wake re-energization.

3:39PM M02.00020 Low-order modelling of wake meandering behind turbines1 VIKKRANT GUPTA, MINPING WAN, Southern University of Science and Technology, Shenzhen — Far-wake regions behind tidal or wind turbines usually have low-frequency oscillations, referred as wake meandering, that cause an increase in turbulence level and thus adversely affect the performance of downstream turbines in an energy farm. We propose a Ginzburg–Landau equation based low-order model for the far-wake region. The model reproduces the main qualitative features of wake meandering: (i) its origin via amplification of upstream structures, (ii) dependence of oscillation frequency on the upstream disturbance amplitude (higher amplitudes lead to lower frequencies), and (iii) shift towards lower frequencies as the wake flow evolves in the streamwise direction. Additionally, the model also predicts the increase in the meandering amplitude and an advancement in its onset with increasing thrust coefficient. To our knowledge, this is the first low-order dynamical system in the literature that models wake meandering. The model coefficients are obtained from the mean flow local stability results that we show correctly account for the changing operating conditions and thus pave way for the prediction of wake meandering features. Its low-order makes it suitable to use inside an energy farm design model, where it can help to mitigate the adverse effects of wake meandering.

1The authors acknowledge the financial support from the US National Science Foundation through Grant No. 1706358 from CBET-Fluid Dynamics Program

1This work was funded by the National Natural Science Foundation of China (Grant Nos. 11672123 and 91752201) and by the Shenzhen Science and Technology Program (Grant No. JCYJ20170412151759222)
Gravitational Effects in Turbulent Two-Phase Heat Transfer in Horizontal Channels

Ilyas Yilgor, Pramod Bhuvanakar, Saedeh Dabiri, Purdue University, DABIRI RESEARCH TEAM — Present work explores heat transfer in horizontal turbulent bubbly flows. High fidelity direct numerical simulations (DNS) are conducted using finite volume and front-tracking methods to analyze the turbulent two-phase heat transfer between two parallel walls with Dirichlet boundary conditions at different temperatures for Eotvos and Archimedes numbers ranging from 0.03-0.6 and 500-10000, respectively. Non-condensable bubbles are present in the flow with a void fraction of 3% and Reynolds numbers ranging from 3000 to 5000. Two-phase simulations are compared to the corresponding single phase simulations with the same flow rate. The improvement in Nusselt number relative to single phase flow is quantified. A critical region where the improvement in heat transfer due to the presence of bubbles equals the convection heat transfer due to a reduced flow rate for a constant pressure gradient is documented. Change of void fraction distributions with gravity, flow rates, average Nusselt numbers and shear stresses are also presented. A range of parameters yielding optimum heat transfer are given.

Temporal dynamics and mode transition in a turbulent Rayleigh-Bnard Convection in a cylindrical domain with a moderate aspect ratio

Yulja Peet, Arizona State University, Philip Sakievich, Sandia National Laboratory, Ronald Adrain, Arizona State University — The current study focuses on dynamics and evolution of large-scale motions in a turbulent Rayleigh-Bnard convection in a cylindrical domain with a moderate aspect ratio of 6.3. We perform Direct Numerical Simulations of the problem with a spectral-element code, and analyze the temporal dynamics of the azimuthal Fourier modes associated with the large-scale motions. Focusing on the first several modes, we document the processes that govern their evolution and interactions, including the in-mode processes, such as fast and slow rotations, as well as inter-mode interactions associated with the mode cessations and transitions. Time scales associated with these processes are analyzed.

Curvature of magnetic field in plasma turbulence

Yan Yang, Minping Wan, Southern University of Science and Technology, Riddhi Bandypadhyay, William Mattheus, University of Delaware, Yipeng Shi, Peking University, Tulasi Parashar, University of Delaware, Quanming Lu, University of Science and Technology of China, Shiyi Chen, Southern University of Science and Technology — Magnetic field lines undergo stretch-twist-fold processes in the presence of turbulence. The curvature field, measuring the tangling of the magnetic field lines, is studied in detail here, using both simulations and observations. The probability distribution function (PDF) of the curvature has distinct power-law tails for both high and low limit values. A central finding is that high curvature co-locates with low magnetic field, which gives rise to the power-law tail of PDF at high curvature. The curvature drift term that converts magnetic energy into flow and thermal energy, largely depends on the curvature field behavior, a relationship that helps to explain particle acceleration due to curvature drift. This adds as well to evidence that turbulent effects most likely play an essential role in particle energization since turbulence drives stronger tangled field configurations, and therefore curvature.

Shear and buoyancy effects in the spatial organization of Stratocumulus clouds

Monica Zamora Zapata, Jan Kleissl, University of California, San Diego — The convective nature of Stratocumulus (Sc) clouds involves the motion of updrafts and downdrafts, driven by cloud-top radiative cooling and surface heat fluxes. Spatially, updrafts reach thicker cloud regions, while downdrafts are found at the thinner or cloud free regions. The balance of surface shear and buoyancy changes the coherent structures (Moeng and Sullivan, 1997). Surface rolls appear with stronger shear, and cells appear when buoyancy dominates. While stronger surface shear increases the cloud fraction of Cumulus clouds (Park et al., 2016), the shear effect on Sc clouds is unknown. Moreover, shear can also occur at the cloud top, where it erodes the Sc cloud from the top (Wang et al., 2012), but the effect on the horizontal spatial properties (aspect and cloud fraction) is unclear. In this work, we study the spatial organization of Sc clouds as a function of shear and buoyancy. We vary the heat flux and wind speed in Large Eddy Simulations (UCLA-LES) of the DYCOMS II RF01 reference case. Cells and rolls are observed at the surface depending on the heat flux and wind speed conditions, with clouds aligning in the direction of the wind for strong surface rolls. On average, the cloud base is more flat and cloud fraction is larger for stronger surface buoyancy.

Short-Wave Instability for Low Reynolds Number Flow over an Inclined Spinning Circular Disk at High Tip-Speed Ratios

Marcus Lee, Tim Colonius, Beverley McKeon, California Institute of Technology — Spin stabilization motivates the study of spinning circular disks for potential application to micro air vehicle design for increased flight robustness. We use a three-dimensional immersed boundary lattice Green’s function method (IBLGF) to simulate flow over a spinning circular disk at angle of attack for Reynolds numbers of $O(10^5)$ and tip-speed ratios up to 3. A short-wave instability emerges in the advancing tip vortex for tip-speed ratios greater than about 1.9. This instability is not present in the non-spinning case and can exhibit frequency lock-in behavior either with the rotation of the disk or with the vortex-shedding instability. Spectral proper orthogonal decomposition (SPOD) of the flow field isolates high-energy modes that help to characterize these instabilities and their coupling.

The simulations presented here used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number CTS120005.
3:49PM M02.00030 Layering, transport and jet formation in rotating, stratified flows with a thermal wind. 1. STEVEN TOBIAS, ADRIAN BARKER, CHRIS JONES, University of Leeds — The Goldreich-Schubert-Fricke (GSF) instability may provide an important contribution to angular momentum transport in planets and stars. We investigate the nonlinear development of the instability, in particular noting the tendency for the transport to be affected by the formation of layers. This is perhaps not surprising as the linear and nonlinear evolution of the equatorial axisymmetric instability is formally equivalent to the salt fingering instability. This is no longer the case in 3D, but we find that the 3D equatorial instability behaves nonlinearly in a similar way to salt fingering. We propose and validate numerically a simple theory for nonlinear saturation of the GSF instability and its resulting angular momentum transport. Away from the equator the nonlinear development is more complicated with layers formed, though these are not perpendicular to the direction of gravity. We conclude by discussing the implications for transport of heat and angular momentum in planets and stars.

1This work was supported using funding from the European Research Council (ERC) under the European Unions Horizon 2020 research and innovation programme (grant agreement no. DG-IVT-786780)

3:50PM M02.00031 Evaporation driven Rayleigh-Taylor instabilities in aqueous polymer solutions. ENDRE MOSSIGE, VINNY SUJA, SAM WHEELER, Stanford University, MEIRBEK ISLAMOV, Columbia University, GERALD FULLER, Stanford University — Understanding the mechanics of detrimental convective instabilities in drying polymer solutions is crucial in many applications such as the production of film coatings. It is well known that solvent evaporation in polymer solutions can lead to Rayleigh-Benard or Marangoni-type instabilities. Here we demonstrate another mechanism, namely that evaporation can cause the interface to display Rayleigh-Taylor instabilities due to the build-up of a dense layer at the air-liquid interface. We study experimentally the onset time \( (t_\text{p}) \) of the instability as a function of the initial polymer concentration \( (c_0) \) and molecular weight. In dilute solutions, \( t_\text{p} \) shows two limiting behaviors. For high diffusivity polymers (low molecular weight), the pluming time scales as \( c_0^{-2/3} \), while in the absence of diffusion, the pluming time scales as \( c_0^{-1} \). Above a critical concentration, \( c_\text{c} \), viscosity delays the growth of the instability, resulting in \( t_\text{p} \) scaling as \( (c_\text{c}/c_0)^{2/3} \). These scaling results are not restricted to polymer solutions or evaporation driven instabilities, but are transferable to other binary systems undergoing gravity driven instabilities.

3:51PM M02.00032 ABSTRACT WITHDRAWN

3:52PM M02.00033 Chaotic and steady regimes of elasto inertial turbulence in 2D channel flows 1. FUQIAN YIN, University of Vermon, USA, JACOB PAGE, RICH KERSWELL, Cambridge University, UK, VINCENT TERRAPON, University of Liege, VICTOR STEINBERG, Weizmann Institute, Israel — Elasto inertial turbulence (EIT) is a non-laminar state of flow occurring in polymer solutions in subcritical and supercritical flows. In 2D subcritical channel flows, the drag increases owing organized polymer dynamics which create flow structures through a backward energy transfer between polymer and the flow. Using viscoelastic direct numerical simulation based on the FENE-P model, we find two main regimes of flow: Chaotic and steady regimes with various variations in between these two bounds. Chaotic flows consist of elongated thin sheets of first normal stress with no particularly defined spacing between the sheets. In steady flow, a peculiar structure, dubbed the super core structure (SCS), emerges. Its existence is controlled by the polymer length and its shape varies with the Weissenberg number. The SCS has exceptional persistence and is speculated to be an exact solution of the flow, as well as a possible connection between EIT and elastic turbulence occurring in inertial-less flows.

1This research was made possible with the support of National Science Foundation CBET-Fluid Dynamics 1805636 and Binaional USA-Israel Foundation 2010145.

3:53PM M02.00034 Sources of flexible wall excitation and wall-pressure fluctuation in turbulent channel flow 1. SREEVATSA ANANTHARAMU, KRISHNAN MAHESH, University of Minnesota - Twin Cities — Structural excitation by turbulent flows result in vibration. We present one-way coupled fluid structure interaction simulations of linear elastic and viscoelastic plates for different material and geometric properties excited by wall-pressure fluctuations generated from Direct Numerical Simulation (DNS) of incompressible turbulent channel flow at \( Re \text{v} \) of 180 and 400. Fluid DNS simulation is carried out in a moving frame of reference using the discrete kinetic energy conserving finite volume method of Mahesh et al. (2004) and the solid simulation is performed in stationary frame of reference using the finite element method. The one-way coupled results are analyzed by a novel framework that represents the plate averaged response spectral density as a double integral of the net source wall-normal cross-spectral density computed for both \( Re \text{v} \) using DNS data. The relative magnitude and phase of the cumulative sources for different frequencies is obtained using spectral proper orthogonal decomposition. Similar technique is applied to analyze the wall-pressure fluctuation spectra. The distribution and the properties of the net sources that contribute to the plate averaged response and wall-pressure fluctuation spectra is discussed for both \( Re \text{v} \).

1This work is supported by the Office of Naval Research.

3:54PM M02.00035 Experimental study of a forced plume in a linearly stratified environment using simultaneous measurements of velocity and density fields. HARISH MIRAJKAR, PARTHO MUKHERJEE, SRIDHAR BALASUBRAMANIAN, Indian Institute of Technology, Bombay India — We present the results of local small-scale measurements of a vertical forced plume ejecting into a linear stably stratified environment, with a stratification strength of \( N = 0.4 s^{-1} \). High-resolution measurements of velocity and density fields were acquired using simultaneous Particle Image Velocimetry (PIV) and Planar Laser Induced Fluorescence (PLIF). The refractive indices of the ambient and the plume fluids were matched to avoid optical aberrations. The energetics of an evolving forced plume were studied by measuring the terms in kinetic energy budget terms, such as mechanical production \( (P) \), buoyancy flux \( (B) \), and dissipation \( (\epsilon) \). The \( P \) and \( \epsilon \) magnitude decreases with increase in the height, while the magnitude of buoyancy flux \( B \), decreases with height and becomes zero at the neutral buoyant layer \( (Z_n) \). Below the neutral buoyant layer, the buoyancy flux, \( B \), shows presence of unstable motions and its sign changes as the plume intrudes into the neutral buoyant layer, indicating stability. The Kolmogorov -5/3 spectral slope is evident in the one-dimensional spatial spectra, indicating the existence of the inertial subrange.
3:55PM M02.00036 Effect of control parameters of traveling-wave blowing and suction on relaminarization phenomenon in fully developed turbulent Taylor-Couette flow

HIROYA MAMORI, The University of Electro-Communications, KOHEI OGINO, KOJI FUKUDOME, Tokyo University of Science, NAOYA FUKUSHIMA, Tokai University, MAKOTO YAMAMOTO, Tokyo University of Science — Direct numerical simulations of turbulent Taylor-Couette flows are performed to investigate the effect of control parameters of a traveling wave control. A traveling wave-like blowing and suction is imposed on inner or outer cylinder walls. The control is aiming the torque reduction effect. A parametric study is conducted to clarify the range of the control effect. A result shows a range of not only torque reduction but also relaminarization phenomenon of turbulent flow. In the inner wall control case, for example, the relaminarization phenomenon occurs, when the wave travels in corotating direction, the wavelength is long, and a wavspeed is faster than the wall velocity of the inner cylinder. We will also discuss the influence of the traveling wave on the Taylor vortex.

3:56PM M02.00037 An affine reconstructed discontinuous Galerkin algorithm for diffusion

YANG SONG, BHUVANA SRINIVASAN, Virginia Tech — In recent years, the discontinuous Galerkin (DG) method has been successfully applied to solving hyperbolic conservation laws. Due to its compactness, high order accuracy, and versatility, the DG method has been extensively applied to extension-convection-diffusion problems. Reliable DG algorithms for hyperbolic terms are well studied. However, an accurate and efficient diffusion solver still constitutes ongoing research, especially for a nodal representation of the discontinuous Galerkin (NDG) method. An affine reconstructed discontinuous Galerkin (aRDG) algorithm is developed in this work to solve the diffusion operator using unstructured NDG method. The proposed numerical approach is computationally efficient, uses minimal storage, and achieves the same order of accuracy as the conventional DG hyperbolic solver. Convergence studies will be presented along with numerical simulations of Rayleigh-Taylor instability growth in the presence of various diffusive mechanisms.

1This work was supported by the US Department of Energy under grant number DE-SC0016515.

3:57PM M02.00038 Analysis of the Flow Over a Sphere Using Direct Simulation Monte Carlo Method in OpenFOAM

TADD YEAGER, DOUGLAS FONTES, MICHAEL KINZEL, University of Central Florida — Continuum assumptions are not valid for some flows such as those found in cases involving atmospheric reentry. For these kinds of problems, the flow field cannot be solved considering the average physical properties (usually used to describe the effects of molecular interactions) as the basic physical processes. The flow field is often characterized by the Knudsen number, which is related to the ratio of the flow’s Mach and Reynolds numbers. Aiming to study of the effects of these relevant dimensionless parameters as they pertain to flow over a blunt body, this paper presents an analysis of rarefied flow over a stationary sphere modeled using the Direct Simulation Monte Carlo (DSMC) method. In these simulations, high subsonic and supersonic flows of air are to be considered and discussed. These cases are simulated using the dsmcFoam solver from OpenFOAM. The preliminary average results of the surface force density and pressure distribution around the sphere surface are consistent with known physics, as are the velocity and momentum fields. Different particle velocities and particle number density should be evaluated to provide a better understanding of interactions between rarefied free stream flow and blunt solid bodies.

3:58PM M02.00039 Stability analysis without numerics: analytic approximations to the pseudospectra of linear operators in fluid mechanics

SCOTT DAWSON, Illinois Institute of Technology — The pseudospectra of a linear dynamical system determine the extent to which, and nature by which, disturbances to the system at a given frequency can be amplified. In practice, pseudospectral analysis (or equivalently, input-output or resolvent analysis) proceeds by computing leading singular values and vectors of the associated discretized operator. This talk will describe a methodology by which the pseudospectra of linear differential operators can instead be closely approximated by prescribed analytic functions. The methodology utilizes results and intuition from wavepacket pseudospectral theory to assume a general form of optimal pseudospectral modes, from which specific mode shapes and growth factors may be obtained by solving a low-dimensional optimization problem for a small number of unknown parameters. This method gives substantial computational savings over standard numerical approaches, and produces accurate results across regimes relevant to real flows. In particular, in such regimes the difference between numerically computed solutions and analytic approximations are typically much smaller than \( \epsilon \) for the \( \epsilon \)-pseudospectrum. We will further discuss connections to other nonmodal stability tools, such as optimal transient energy growth analysis.

3:59PM M02.00040 Scale dependence of entrainment bubble size distribution in free-surface turbulence

XIANGMING YU, KELLI HENDRICKSON, DICK YUE, Massachusetts Institute of Technology — Air entrainment by free-surface turbulence plays important roles in both natural processes and engineering applications. We consider the size spectrum of surface entrained bubbles under strong free-surface turbulence (SFST) and develop a physical/mechanistic model for the entrainment bubble-size spectrum per unit interface area \( \mathcal{N}_e(r) \). The model defines the spectrum dependence on gravity \( g \), surface tension \( \sigma / \rho \), and turbulence dissipation \( \epsilon \), and obtains two distinct entrainment regimes separated by bubble-size scale \( r_0 \). From the model we show that \( r_0 = r_c = \frac{1}{2} \sqrt{g \sigma / \rho g} \), the capillary length scale, and not the Hinze scale \( r_H \) as is generally assumed. For an air-water interface and earth gravity, \( r_c \approx 1.5 \text{mm} \). We confirm the theoretical model by high-fidelity, two-phase, volume-conserving direct numerical simulations (DNS) of a canonical SFST flow. We will present: (1) the respective power-laws of the two regimes; (2) the value \( r_0 = r_c \neq r_H \); (3) the scaling of \( \mathcal{N}_e \) with \( g \sigma / \rho \) and \( \epsilon \); and (4) confirmation of the \( \epsilon \)-- \( r \) entrainment regime map predicted by the model.

1Office of Naval Research

4:00PM M02.00041 Investigation of the effect of evaporating ocean spray on the air-sea heat fluxes in high-wind conditions

YEVENII RASTIGEJEV, North Carolina Agricultural and Technical State University, SERGEY A. SUSLOV, Swinburne University of Technology, Australia — We have studied the effect of evaporating ocean spray droplets of typical sizes on air-sea heat fluxes in a marine atmospheric boundary layer (MABL) with various vertical profiles of air temperature and moisture and values of turbulence intensity in high-wind conditions. We have found that the vertical latent and total heat fluxes are strongly enhanced by large spray droplets with radii 0.5mm because their presence results in steep vertical gradients of moisture and temperature in a MABL. The effect of small droplets on the total heat flux is not as profound: fine spray primarily redistributes thermal energy between its latent and sensible components. We have shown that spray affects the turbulent kinetic energy (and thus the intensity of the vertical turbulent transport) mostly mechanically (by altering the vertical distribution of mass density of the air-spray mixture) rather than thermodynamically (by changing vertical distributions of the air temperature and moisture). We have compared the dependence of the total vertical heat flux on the wind speed produced by the current model with the observation data that show seemingly anomalous growth of the vertical heat flux with the wind speed. We showed that this may be explained by the presence of ocean spray in a MABL.

1YR acknowledges support by a grant from the National Science Foundation, U.S.A. under Award No. AGS-1832089
4:01PM M02.00042 Clustering and settling of snow particles in atmospheric turbulence

This work is supported by NSF-AGS-1822192.

4:02PM M02.00043 ABSTRACT WITHDRAWN

4:03PM M02.00044 ABSTRACT WITHDRAWN

4:04PM M02.00045 Energy losses induced by channel-spanning brush accumulations

This project has received funding from the European Union’s Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement WoodJam No. 745348.

4:05PM M02.00046 Experimental model for visualization of the isotherm patterns conformed in the steam chamber in SAGD oil recovery

This research was made possible by a grant from the Gulf of Mexico Research Initiative.

4:06PM M02.00047 A Tale of Two Droplets: Wetting Phenomena on Oil-infused Surfaces

REG & CICE Fundings at Lamar University

4:07PM M02.00048 Reduced 1D population dynamics model for inflow size distribution in LES of oil droplet plumes

This research was made possible by a grant from the Gulf of Mexico Research Initiative.
4:08PM M02.00049 Oil Droplet and Sediment Suspension in Laboratory-Scale Stommel Retention Zones, CARLOWEN SMITH, ZONGZE LI, University of South Florida Department of Mechanical Engineering, ANDRES TEJADA-MARTINEZ, University of South Florida Department of Civil and Environmental Engineering, DAVID MURPHY, University of South Florida Department of Mechanical Engineering — Langmuir supercells are helical wind- and wave-driven circulations in the ocean with alternating regions of upwelling and downwelling that extend to the full depth of the water column. These flows can trap suspended materials in sub-surface regions of turbulent vertical motion known as Stommel Retention Zones (SRZs), an effect that can specifically impact oil-particle aggregation following an oil spill. We present a laboratory facility recreating some aspects of SRZs and its characterization using PIV. The experimental facility consists of a 10.20.5 m tank in which a shear stress is applied on the side walls using conveyor belts, resulting in a counterrotating vortex pair with either a central downwelling or upwelling region of variable strength and turbulent kinetic energy. Mean up/downwelling flow speeds range from 0.05-0.2 m/s, where turbulent kinetic energy ranges over two orders of magnitude. The facility is used to study oil and particle dynamics within these zones of turbulent retention. Positively buoyant oil droplet and negatively buoyant sediment particle trajectories and spatial concentration fields are quantified, and droplet retention is then compared with sediment suspension, yielding information about the expected efficiency of oil-particle aggregation.

4:09PM M02.00050 3D PTV in a spray cloud from wave impact with wind interaction, REYNA RAMIREZ DE LA TORRE, ATLE JENSEN, UiO — Marine icing is a phenomenon of growing importance due to the increase of marine traffic in the Arctic sea. An interesting example is the ocean spray freezing on top of vessels. However, its dynamics is still to be understood. These type of studies are important for the safety of people, ships and installations that operate in the Arctic environment. It has been reported that sea spray formation is caused mainly by wave impact and wind. Therefore, an experimental model of this phenomenon was created to describe the trajectories of droplets in the spray cloud generated by wave impact and wind. The experiments were developed in the wave plume of the Hydrodynamics Laboratory in the University of Oslo. Breaking waves were generated, while a fan was used to produce wind on top of the waves. Droplet trajectories were reconstructed by 3D Particle Tracking Velocimetry. Using this setup we studied the dynamics of spray clouds with different wind conditions and compared the experiments with simulations. The overall goal is to develop solutions to reduce the icing of Arctic structures.

4:10PM M02.00051 ABSTRACT WITHDRAWN —

4:11PM M02.00052 Digital Aerodynamics, FLAVIO NOCA, HEPIA / HES-SO University of Applied Sciences, GUILLAUME CATRY, WindShape — Conventional wind tunnels have a limited number of fans. They are generally programmed to all turn at the same speed, thus generating a uniform and permanent flow. We have developed a technology to shape the morphology of wind in space and time. It is based on a large number of fans (wind pixels), which are distributed arbitrarily in space and can be modulated individually in time. We will provide preliminary experimental measurements on the correlation between a given fan-speed distribution and the resulting flow pattern in the test section using PIV and multihole pressure probes.

4:12PM M02.00053 ABSTRACT WITHDRAWN —

4:13PM M02.00054 ABSTRACT WITHDRAWN —

4:14PM M02.00055 Two-dimensional partially-ionized magnetohydrodynamic turbulence, SANTIAGO BENAVIDES, GLENN FLIERL, Massachusetts Institute of Technology — Ionization occurs in the upper atmospheres of Hot Jupiters and in the interiors of Gas Giant Planets, leading to Magnetohydrodynamic (MHD) effects which couple the momentum and the magnetic field, thereby significantly altering the dynamics. In regions of moderate temperatures the gas is only partially ionized, which leads to interactions with neutral molecules. To explore the turbulent dynamics of these regions we utilize Partially-Ionized MHD (PIMHD), a two-fluid model — one neutral and one ionized — coupled by a collision term proportional to the difference in velocities. Motivated by planetary settings where rotation constrains the large-scale motions to be mostly two-dimensional, we perform a suite of simulations to examine the parameter space of 2D PIMHD turbulence and pay particular attention to collisions and their role in the dynamics, dissipation, and energy exchange between the two species. We arrive at, and numerically confirm, an expression for the energy loss due to collisional heating in both the weakly and strongly collisional limits, and show that, in the latter limit, the neutral fluid couples to the ions and behaves as an MHD fluid.

4:15PM M02.00056 Multi-lens stereo reconstruction of the free surface waves in wave basin

1, HUA LIU, QIAN WANG, Shanghai Jiao Tong University, KEY LABORATORY OF HYDRODYNAMICS (MINISTRY OF EDUCATION OF CHINA) TEAM — A Multi-lens stereo reconstruction approach is developed to measure the free surface deformation of water waves in a wave basin. A massive of tiny granule served as the thin film floating on the free surface ensures the function of the common multi-lens stereo imaging technique in reconstructing the wave surface. The granule film and the highbrightness projector produces distinct pattern features on the free surface. The effect of the granule film on the wave propagation is checked which turns out that there is little influence on the wave propagation. Comparing the surface elevation computed from the reconstructed wave surfaces with the wave gauge data, a good agreement is found. The developed multi-lens stereo reconstruction approach is applied in investigating the propagation of a solitary wave over a submerged plate in a wave basin. The 3D deformation of the free surface with the high precision and efficiency for the considerable measuring area will be presented.

1This study is supported by the National Natural Science Foundation of China (Grant Nos. 11632012 and 14861144024).
3:20PM M03.00001 Long-term low shear of a waxy potato starch paste produces a highly viscous gel by formation of intermolecular double-helices.  FANG FANG, Whistler Center for Carbohydrate Research and Department of Food Science, Purdue University, XIAO ZHU, Information Technology at Purdue (ITaP), Purdue University, MARIO MARTINEZ, Whistler Center for Carbohydrate Research and Department of Food Science, Purdue University, OSVALDO CAMPANELLA, Whistler Center for Carbohydrate Research, Purdue University, Department of Food Science and Technology, The Ohio State University, BRUCE HAMAKER, Whistler Center for Carbohydrate Research and Department of Food Science, Purdue University — Our group recently reported that waxy potato and corn pure starch amylopectin pastes undergo a shear-thickening behavior at a low shear rate range of 5 to 25 s⁻¹, and that the effect did not occur in waxy rice starch. Here, we show that gelatinized potato amylopectin subjected to prolonged shear at 20 s⁻¹ for 24 h at 5°C produced a highly viscous gel with 5x more double-helices than without shear. Shear-induced aggregates formed within 20 min. Double-helices melted between 40 to 75°C leading to a total loss of gel elasticity and a steep decrease in G’, indicating that the retrogradation process occurred in part among amylopectin molecules (i.e. intermolecularly). Thus, a new phenomenon is reported whereby waxy potato amylopectin with long linear chains forms double-helical aggregated structures in the presence of long-term low shear that dramatically affects material properties.

3:21PM M03.00002 Numerical and experimental investigation of fluid flow in tapered orifices for needle-free injectors.² YATISH RANE, JEREMY MARSTON, Texas Tech University — Transdermal drug delivery using spring-powered jet injection has been studied for several decades, and is an attractive option for delivery of highly viscous and non-Newtonian fluids. In particular, third-generation vaccines such as DNA-vaccines have shear-thinning behaviour, which dictates the need to study the influence of fluid rheology in jet injection. Here, numerical simulations are performed with steady state flow and turbulence modelling based on the system Reynolds number at the orifice to generate characteristic curves of dimensionless pressure drop (Euler number) versus generalized Reynolds number. The results are experimentally validated for a given geometry over a wide range of Reynolds numbers (10¹ - 10⁴), and we find shear-thinning (Carreau) fluids with high zero-shear viscosities can be injected due to the presence of high shear (10⁶) regions near the orifice. In addition to fluid rheology, the orifice geometry (e.g. conical, multi-tier tapers and sigmoid contraction) is varied to study boundary layer thickness, which also affects jet collimation. Ultimately, our results indicate there may be optimal geometries for creating jets to target specific tissue depths.

²National Science Foundation, Inovio Pharmaceuticals.

3:22PM M03.00003 ABSTRACT WITHDRAWN –

3:23PM M03.00004 Acoustic Streaming in Confined Liquid Subject to High Intensity Focused Ultrasound (HIFU) Exposure³ GHADEM OWEIS, HUSSEIN DAOUD, RASHA SEBLANY, Mechanical Engineering, American University of Beirut — HIFU is an ex-corporeal therapeutic modality for the non-invasive treatment of tumors. A HIFU transducer emits spherically focused ultrasonic energy unto the target tissue deep under the skin, which is consequently absorbed and turned into heat, causing thermal ablation. Non-thermal effects can be present too, including cavitation bubble formation, radiation force, and acoustic streaming. The liquid streaming patterns were elucidated in previous HIFU investigations in an infinite medium. The aim of this study is to probe the effect of fluid confinement size on acoustic streaming. Two cubic water cuvettes of 5 mm, and 20 mm confinements were placed in an acoustic-coupling water bath at the geometric focus of the HIFU transducer (1.6 MHz, f ≠ 1, 60 mm). A single 30 ms HIFU pulse exposure was used. PIV was implemented to measure the centerline streaming field (radial and axial components) at the end of the acoustic emission (avg. of 100 repeats). Significant differences were found in the flow patterns between the small (5 mm) and large (20 mm) confinements. The streaming velocity levels were also dramatically different. In conclusion, fluid confinement has a very important effect on the streaming flow from HIFU.

³The data for this study were acquired for the course project (MECH 609, Exp. Fluid Dynamics, AUB, 2017)

3:24PM M03.00005 Could an implantable sensor both monitor lung flow and generate power?⁴ LUCY FITZGERALD, LUIS LOPEZ RUIS, JIANZHONG ZHU, JOHN LACH, DANIEL QUINN, University of Virginia — The rise of Smart Health has led to implantable healthcare devices that can diagnose and monitor diseases in real-time. Some diagnoses are based on fluids in the body, like asthma. A key challenge of current implantables is that they are difficult to power and require surgeries to replace batteries. We show that a piezoelectric flow sensor could monitor a fluid flow and simultaneously power itself from that flow. The effectiveness of this sensor/harvester depends on flow properties. It therefore demands advanced models that capture tradeoffs between sensing fidelity and harvesting potential. To develop these models, we built a platform for testing the sensing/harvesting capability of piezocantilevers in airflows modeled after human breathing. We found that oscillating voltage on the piezocantilever can both charge a capacitor and map to the amplitude and frequency of the breath. We explored how well models can predict harvesting and sensing potential based on breath type. These models open up a broad range of applications that we are exploring, including a smart stent that alerts patients to obstructions or dislodging. How these models scale with flow speed/direction, turbulence intensity, and cantilever size offer design ideas for sensing/harvesting in other bodily channels.

3:25PM M03.00006 Physics-Informed Deep Neural Nets for Super-Resolution, Denoising, and Velocity-Aliasing Correction of 4D-Flow MRI ROSHAN D’SOUZA, University of Wisconsin-Milwaukee, MOJTABA FATHI, ISAAC PEREZ-RAYA, University of Wisconsin - Milwaukee — 4D-Flow MRI is a non-invasive method for measuring blood velocities in the human vascular system. It suffers from issues such as low spatio-temporal resolution, acquisition noise, and velocity-aliasing artifacts. Here, we present a novel method based on physics-informed Deep Neural Nets (DNNs) for super-resolution, denoising, and velocity-aliasing correction of 4D-Flow MRI. The inputs are the sparsely-sampled 4D-Flow MRI images and a rough geometry mask. The loss function used to train the physics-informed DNN contains terms for volume-averaged data fidelity and point-wise flow physics. Noise is reduced by using an averaging filter on the input data as well as the DNN predictor. Velocity aliasing is handled by computing data fidelity in the velocity encoding space. The trained DNN can be sampled at any desired resolution to generate noise-free flow images without velocity aliasing. Preliminary tests on CFD-derived 2D synthetic data shows that the method is able to automatically address all the aforementioned issues with normalized-root-mean-squared-error in velocity less than 4.8 percent and direction error less than 0.096 when compared to the reference CFD.
3:26PM M03.00007 MRI-based modeling of CSF flow in the spinal canal1. JENNA J LAWRENCE, WILFRED COENEN, University of California San Diego, CANDIDO GUTIERREZ-MONTES, Universidad de Jaen, ANTONIO L SANCHEZ, University of California San Diego, CARLOS MARTINEZ-BAZAN, Universidad de Jaen, KEVIN KING, Huntington Medical Research Institutes, VICTOR HAUGHTON, University of Wisconsin Madison, JUAN C LASHERAS, University of California San Diego — The oscillatory flow of cerebrospinal fluid (CSF) in the subarachnoid space of the spinal canal is driven by the intracranial pressure fluctuations associated with cerebral blood flow and by the thoracic pressure fluctuations associated with the respiratory cycle. We have previously derived simplified flow models by exploiting the slenderness of the subarachnoid space and the limited deformation of the dura membrane. Application of these models to a specific human subject requires knowledge of their spinal-canal anatomy and of both their spinal-canal and cranial-cavity compliance. We show how this specific information can be extracted from high-resolution magnetic resonance (MR) imaging of the anatomy, along with MR phase-contrast flow measurements of venous and arterial blood flow at the C2 level and of CSF flow at several transverse sections along the spinal canal. We then show how the resulting subject-specific model can be used to predict steady bulk motion and drug-dispersion rates.

1National Science Foundation, Award Number 1853954

3:27PM M03.00008 In Vitro Assessment of Cycle to Cycle Flow Variability in Intracranial Aneurysms using Radial 4D Flow MRI and Tomographic PIV., RAFAEL MEDERO, KATRINA RUEDINGER, DAVID RUTKOWSKI, KEVIN JOHNSON, ALEJANDO ROLDN-ALZATE, University of Wisconsin-Madison — Intracranial aneurysm rupture has been related with aneurysm geometry, and high flow activity. 4D Flow MRI has been shown to be a feasible imaging technique for assessing hemodynamics in different vascular territories. However, one of its limitations is the need to average several cardiac cycles to obtain a complete data set, causing a smoothing of the velocity profiles and errors in areas with non-laminar flow. Additionally, it requires prospective determination of the velocity encoding setting, restricting the range of velocities acquired. Furthermore, the need for reasonable scan times can lead to limits in spatial resolution, which motivates the development of improved MRI sequences such as radial acquisitions. In this study, velocities acquired with radial 4D Flow MRI where compared to tomographic PIV using a patient-specific intracranial aneurysm in-vitro model under pulsatile flow. Velocity data from multiple time points within a group of 10 cardiac cycles acquired with tomo-PIV were compared pixel-to-pixel, and averaged velocity data was compared between methods. Statistically differences were found between velocities measured with tomo-PIV at peak systole and end diastole. However, good agreement was seen when comparing 4D Flow MRI with the average time points.

3:28PM M03.00009 An experimental study of pulsatile flow over rectangular sidewall cavities, RUIJANG ZHANG, BENJAMIN EICHHOLZ, YAN ZHANG, North Dakota State University — Open cavity flow is a classic benchmark fluid dynamic model that has been extensively studied over the past decades. The existence of the free shear layer causes variations of vortex structures and flow stagnations inside the cavity in different flow regimes. However, how the flow pulsatility affects the vortex dynamics of the cavity flow is still not fully understood. Such question is of critical importance to many biological flow phenomena, such as blood past the brain aneurysms and left atrial appendage blood flows. The goal of this study is to reveal the flow characteristics of two simple rectangular sidewall cavity models under physiologically-relevant pulsatile flow. Cavities with two depth-to-width ratios were studied. Flow waveforms were generated using a programmable pulsatile pump to mimic the heart functions. Phase-locked PIV were conducted to study the cyclic variation of vortex flow structures inside and across the cavity. The velocity and vorticity fields were analyzed and found to significantly vary at different peak Reynolds numbers, Womersley numbers, and pulsatility indices. This study represents a systematic experimental effort towards pulsatile flow over a standard cavity model, which could serve as a benchmark for future computational simulations.

3:29PM M03.00010 Elucidating Left Ventricular Hemodynamics and Aggregation Zones Using Platelet-focused Lagrangian analysis1, VENKAT KESHAV CHIVUKULA, FANETTE CHAS-SAGNE, JENNIFER BECKMAN, CLAUDIUS MAHR, ALBERTO ALISEDA, University of Washington — Left Ventricular Assist Devices (LVAD) have improved significantly over the last three decades and its use has expanded beyond the original Bridge-to-transplant indication. Thromboembolic complications, however, have not decreased in frequency or severity despite the advances in pump design. We investigate unfavorable hemodynamics in the left ventricle (LV) of a HF patient implanted with an LVAD. High-fidelity computational fluid dynamics are used to quantify thrombogenicity in the LV for several implantation configurations. Platelet Lagrangian tracking characterize the mechanical stimuli along individual trajectories, including residence time and shear stress history. Rigorous statistical analysis reveals recirculation zones inside the LV where platelet aggregation and thrombus formation can occur. PIV in a flow phantom implanted with a real-world LVAD provides validation. Clinically relevant parameters and patient management strategies are considered. A mathematical valve opening that encourage the patient’s ventricular contraction are assessed to optimize intraventricular flow patterns. Risk stratification is demonstrated to develop strategies that minimize stroke risk and have the potential to improve patient outcomes.

1This work was partially supported by an AHA postdoctoral fellowship 16POST30520004 and the Locke Trust Foundation of the University of Washington

3:30PM M03.00011 Hemodynamics of the left heart with physiologic and pathologic mitral valve: the chordae tendinae effect, ROBERTO VERZICCO, Uniroma2, UTwente, GSSI, VALENTINA MESCHINI, Uniroma2, FRANCESCO VIOLA, UTwente — One of the advantages of computational engineering is the possibility to vary one factor of a system, while leaving the others unchanged, and to assess its effect. In cardiovascular flows this strategy allows to selectively investigate modifications of the intraventricular hemodynamics produced by myocardial tissue remodeling or pathologies. In this study, we rely on our in-house multi-physics numerical model that dynamically couples electrophysiology, tissue mechanics and hemodynamics in physiological and pathological conditions to carry out direct numerical simulations of the left heart dynamics. Here, we present the effect of loose or broken chordae tendinae of the mitral valve on the ventricular pumping efficiency in terms of cardiac output, valve regurgitation and large scale flow structures. The results are seen to agree with the available clinical data, thus suggesting that this computational tool could be used to predict the effects of a valve sparing procedure and to improve the outcome of a surgical interventions.
3:31PM M03.00012 Computational Hemodynamics of Prosthetic Aortic Valves with Application to Continuous Monitoring of Valve Function1. SHANTANU BAILLOOR, JUNG-HEE SEO, Johns Hopkins University, HODA HATOUM, LAKSHMI PRASAD DASI, Ohio State University, RAJAT MITTAL, Johns Hopkins University, JOHNS HOPKINS UNIVERSITY TEAM, OHIO STATE UNIVERSITY TEAM — Transcatheter heart valves suffer from complications such as leaks, thrombosis, endocarditis etc. Of these, sub-clinical or clinical thrombosis, even if resolved by anti-coagulation therapy, impacts the long-term durability of the valve. Technology to help avoid these complications and detect them very early is key towards advancing heart valve therapy. A small number of wireless pressure micro-sensors mounted at strategic locations on the valve frame could enable continuous monitoring and alerting to very early stages of thrombosis or other complications, as well as to guide anti-coagulation therapy or other clinical management. We employ hemodynamics simulations of transvalvular flow in a canonical model of the aorta with a transcatheter valve and determine optimal sensor configurations for discriminating between healthy leaflets and those exhibiting reduced mobility. By applying machine-learning based techniques to a large cohort of in-silico aorta models, we demonstrate that a small number of in-situ sensors can effectively predict early-stage leaflet abnormalities.

1The authors acknowledge support from NSF Grant CBET-1511200, and NSF XSEDE Grant TG-CTS100002.

3:32PM M03.00013 A novel 1D flow model for transient FSI simulation of aortic valve , HAOXING LUO, YE CHEN, Vanderbilt University, LUO LAB TEAM — In contrast with costly 3D flow simulations, simple aortic flow models are much more efficient and have applications in non-invasive clinical flow measurement as well as design optimization of prosthetic valves. Existing simple flow models are typically based on the old Bernoulli principle. In this work, we have developed a novel one-dimensional (1D) unsteady flow model based on the momentum and mass conservation equations, which takes into consideration of both valve movement and pressure loss through the valve. We couple this simplified flow model with a 3D FEM model of the thin-leaflet aortic valve as a 1D/3D hybrid model and perform FSI simulations for the valve of different bending rigidity. The nonuniform pressure distribution on the leaflets is included in the simulation. The full 3D FSI model that simulates the 3D pulsatile flow fully coupled with the same valve is used to assess the performance of the simplified flow model. The results show that the hybrid model is able to capture reasonably well the 3D deformation sequence of the leaflets, the valve opening area, and the flow rate, especially in the cases of low bending rigidity.

3:33PM M03.00014 Venous valve dynamic behavior and function: a computational investigation. MATTHEW BALLARD, PARKER ELLIOTT, Saint Martin’s University — To fulfill their role of returning blood from the body back to the heart, veins often need pumping beyond that provided by the heart. This is especially true where blood must be raised vertically a significant distance against gravity, such as from the deep veins of the legs. Thus, many veins contain a series of “venous valves” which open and close with pressure fluctuations to allow flow only in the direction back toward the heart. These valves enable veins to act as a series of “miniature hearts” that provide the requisite pumping effect. Under certain conditions including flow stasis associated with sitting still, long airplane flights and surgery, venous valves can become blocked through thrombus formation (deep vein thrombosis, or DVT), causing insufficient blood flow. Thrombi can even break free and become lodged in the lungs as a deadly pulmonary embolism (PE). We use a three-dimensional, fully coupled fluid-solid model based on the lattice-Boltzmann and lattice spring models to investigate the behavior of a viscous fluid and venous valves in a section of vein. We study the dynamics of venous valves and assess the effect of valve morphology on fluid transport. Further, we study flow in the valve region with a focus on understanding its effect on development of disease. This work increases our understanding of venous valve behavior and the resulting flow conditions, with possible applications in evaluating patient risk for DVT and in designing prosthetic venous valves.

3:34PM M03.00015 Can you hear better if you’re lopsided? Tympanic asymmetry may enhance hearing in a parasitoid fly1. MAX MIKEL-STITES, PAUL MAREK, ANNE STAPLES, Virginia Tech — *Ormia ochracea* is a parasitoid fly endemic to the Americas. Gravid females respond phonotactically to calls of their male *Gryllidae* cricket hosts. Astonishingly, *Ormia* can locate the source of sound equal to that much proucerates, in spite of their small size, which should prohibit this level of precision because of fundamental constraints imposed by the physics of sound propagation (Mason et al., Nature, 2001). Miles et al. demonstrated that *Ormia* is capable of resolving nanosecond time differences due to a direct mechanical coupling of the flies tympanic membranes (Miles et al., J Acoust Soc Am, 1995). This mechanical coupling increases the interaural time delay (ITD) between the tympana, thus enhancing the flies sound localization precision. Here, we introduce an asymmetry in tympanic area into the mathematical model provided by Miles et al. and demonstrate that an asymmetry of less than 10% between the left and right tympanal areas can more than double the ITD. We further present initial measurements of 44 *Ormia* tympana that demonstrate an average asymmetry in tympanal area of approximately 5%.

1Thank you to undergraduate Maddie Hellier for all her work measuring the O. ochracea tympanal membranes!

3:35PM M03.00016 Thumbs up! Bird’s thumb induces leading-edge vortex during slow gliding flight1. THOMAS LINEHAN, KAMRAN MOHSENI, University of Florida — The alula, or bird’s thumb, consists of a small set of feathers stationed near the bird’s wrist that protract from the wing during slow flight to enhance lift. The recent discovery of leading-edge vortices (LEVs) on the thin hand-wings of some birds suggest that the alula may play a role in LEV formation. Using stereoscopic-digital particle image velocimetry we measured the flow over a model wing with and without protracted alulae in a wind tunnel and made volumetric reconstructions of the three-dimensional vortex flow. We found that the alula induces and stabilizes a robust conical leading-edge vortex (LEV) that sweeps across the outer wing and smoothly merges with the tip vortex. LEV formation is the result of the alula scraping spanwise vorticity from the leading-edge shear layer and inducing its roll-up. The subsequent stabilization of the LEV is the result of root-to-tip spanwise flow in the LEV core of magnitude greater than 80% of the freestream value. In essence, the protracted alula, mimicking a canted flap, is a clever way of inducing and stabilizing a LEV on a steadily translating wing inclined to the flow at high angles. These results grant new insights into the intelligent design of the modern bird wing and have important implications for aircraft flight control.

1The authors gratefully acknowledge funding from the National Science Foundation, Award 1805776.
3:36PM M03.00017 Wake Dynamics of Bat Flapping , VAIBHAV JOSHI1, RAJEEV JAIMAN, The University of British Columbia — Natural selection has evolved the geometry as well as mechanical properties of wings of a bat to achieve better flight performance, maneuverability and agility. The highly anisotropic and deformable membranes of the flapping wing and complex kinematics make their study more imperative for bio-inspired aerodynamic applications such as micro-air vehicles (MAVs). The current study is a first step towards understanding such complex flapping dynamics using a flexible multibody fluid-structure interaction framework. We aim to study the effect of the flexibility or compliance of the wing on the vortex dynamics and flight performance during hovering of the bat. We find that for the same power input to the flapping wing, the flexible compliant wing has better flight performance compared to its rigid counterpart. Moreover, the vortex structures generated supply more vorticity to the vortex ring patterns in the flexible wing leading to its large amplitude of deformation and unsteady lift coefficient.

1Membership Pending

3:37PM M03.00018 Hydrodynamic thrust generation by honeybee (Apis mellifera)1 , CHRIS ROH, MORTEZA GHARIB, California Institute of Technology — In our previous studies, we reported honeybee’s locomotion at the air-water interface. Their ventrally wetted wings beat at high frequency (30-220 Hz), which propel them forward. Honeybee’s locomotion on a water surface uses added mass as the dominant hydrodynamic force. Disregarding other forces, an added mass force associated with idealized sinusoidal wing kinematics is modeled. The resulting body movement shows good agreement with the experimentally measured body motion. Furthermore, body kinematics predicted based on the experimentally measured flow field under the mechanical model wing also shows similar locomotive pattern.

1This material is based upon work supported by the National Science Foundation under Grant No. CBET-1511414; additional support by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1144469

3:38PM M03.00019 ABSTRACT WITHDRAWN –

3:39PM M03.00020 Bioinspired Passively Actuated Microflap Surface for Improving Airfoil Performance1 , SEAN DEVEY, CHRIS JARMON, AMY LANG, PAUL HUBNER, The University of Alabama — Flow separation acts to limit the efficiency of aerodynamic systems. A novel dynamic surface is proposed as a mechanism to limit flow separation. This surface is derived from the flank skin of the shortfin mako shark, which has been proven effective at limiting flow separation in adverse pressure gradient flows. An array of passively actuable “microflaps” mimics the geometry and flexibility of mako flank denticles. It is hypothesized that these microflaps will respond to local reversing flows to passively enforce a selective flow direction and increase mixing within the boundary layer to limit flow separation. This surface has been produced with additive manufacturing and incorporated into the upper surface of a NACA 0012 airfoil. Low speed wind tunnel testing (Re ~2e5) of this airfoil is in progress. An increase in maximum lift and delay of stall is expected due to limiting of flow separation by the microflap array.

1The work was supported by REU Grant 1659710.

3:40PM M03.00021 Fluid dynamics of nutrient exchange through the branched arrays of sea fans and jellyfish oral arms , LAURA MILLER, University of North Carolina at Chapel Hill — Numerous small organisms that swim, fly, smell, or feed in flows at the mesoscale, where inertial and viscous forces are balanced, rely on branched, bristled and hairy structures. Such mesoscale structures can augment particle capture and nutrient exchange by moving in a manner to transition from acting as solid surfaces to leaky/porous rakes at Reynolds numbers close to one. Although mesoscale flows have been studied in many organisms, the fluid dynamic mechanisms underlying the leaky rake to solid plate transition remain unclear. A detailed understanding of how this leaky-to-solid transition affects chemical exchange and particle capture in mesoscale filtering, where advective and diffusive transport rates are nearly balanced, also remains unavailable. In this presentation, flow visualization and computational fluid dynamics will be used to quantify the fundamental fluid dynamics of biological filtering arrays in this regime. Two types of marine invertebrates will be examined for understanding mesoscale biological filters: 1) upside-down jellyfish that use bell pulsations to filter particles within 3D bristled oral arms, and 2) sea fans that are branched into approximately 2D sheets.

3:41PM M03.00022 CFD Time Machine: Using CFD to Understand the History of Long Extinct Swimmers , NICHOLAS HEBDON, KATHLEEN RITTERBUSH, University of Utah Geology and Geophysics, YUNJII CHOI, Jacobs Engineering Group — Ammonoids are a group of cephalopods that were found in oceans worldwide for nearly 300 million years. These organisms once swam freely in the ocean and were as abundant as fish are today but went extinct 65 million years ago in the same extinction that killed the dinosaurs. In this study, we use Computational Fluid Dynamics model (Ansys V18.0) to study the locomotion of these now extinct animals. The fossil record shows that ammonoids underwent numerous periods of evolutionary boom and bust. During these cycles, ammonoids with a planispiral shell type often show clear shifts in their shell morphology, strongly emphasizing traits such as lowered cross-sectional area or increased coiling exposure. However, because there are no modern members of the group and there is no soft tissue preservation, it has been difficult for researchers to understand the implications of these shifts on their swimming pattern. We used CFD to study the hydrodynamic impacts of their morphological transformation, in terms of drag and lift. Incremental changes in shell shape across three common shape characters were studied: Shell inflation (total shell width), Whorl Expansion (the rate at which coil diameter increases, and Umbilical Exposure (the ratio of exposed coiling to total diameter). Our results show that drag and lift are sensitive not only to the particular parameter being changed but also that they are non-uniformly sensitive to the direction (increasing or decreasing) and magnitude of how that parameter is being changed.
3:42PM M03.00023 Effect of viscosity on Lingulodinium Polyedrum micro swimmer motility and properties¹, LOURDES MNICA BRAVO ANAYA, Laboratoire Interdisciplinaire de Physique (LiPhy), HUGUES BODIGUEL, FRDRIC PIGNON, Laboratoire Rhologie et Procds, MARVIN BRUN-COSME-BRUNY, PHILIPPE PLEYLA, SALIMA RAFAI, Laboratoire Interdisciplinaire de Physique (LiPhy), LABORATOIRE INTERDISCIPLINAIRE DE PHYSIQUE (LIPHY) COLLABORATION, LABORATOIRE RHLOGIE ET PROCDS COLLABORATION — Lingulodinium Polyedrum (LP) microalgae is a marine protist, belonging to the dynoflagellata gender that emits bright blue light flashes once submitted to a shear stress. It carries an equatorial flagellum and a longitudinal one, allowing the swimming. In this work, we studied the effect of viscosity on LP microswimmer motility and bioluminescence properties through rheological measurements and a bright field microscopy imaging. The microalgae velocity dependency on the viscosity of the medium was measured in the presence of different concentrations of selected polymers. Xanthan was selected due to its shear-thinning behavior and a hydrolyzed polyacrylamide was also chosen due to its Newtonian behavior at concentrations lower than C*. These polymers were found to be biocompatible and stable in seawater and in presence of LP. The swimming velocity of LP microswimmer was found to be inversely proportional to the suspension viscosity for both polymers, suggesting that the force needed to stop the swimming LP cell remains constant. Finally, it was observed that the applied shear rate necessary for LP microalgae to emit bioluminescence decreases while increasing the suspension viscosity.

¹TEC21 Grant

3:43PM M03.00024 Reversal of Flagellar Wave Propagation Is Controlled by Proximal to Distal Asymmetry in Molecular Motor Dynamics¹, FENG LING, YI MAN, EVA KANSO, University of Southern California — The '9+2' axoneme of the motile cilium/flagellum is an important cellular structure that is highly-conserved among eukaryotic cells. Asymmetries of dynein motors along the flagellum have been identified in a number of organisms, and they have been specifically linked to the reversal in the direction of flagellar wave propagation in certain trypanosomes. Trypanosomes are a class of single-celled parasites that are known to switch their flagellar beating between tip-to-base and base-to-tip waveforms. In this talk, we analyze cilia oscillations and direction of wave propagation in the context of a known geometric feedback model. We introduce proximal to distal asymmetry in the molecular motor dynamics, consistent with recent experiments on trypanosomes. We show that the experimentally-observed reversal of wave propagation is only achievable when sliding-control feedback is dominant. We conclude by commenting on the implications of these results to flagellar waveforms in other organisms, and the feasibility of a universal geometric feedback mechanism for explaining the diverse waveforms in cilia oscillations.

¹National Science Foundation (NSF) INSPIRE grant NSF 1608744, Office of Naval Research (ONR) grant N00014-17-1-2062, Army Research Office (ARO) grant W911NF-16-1-0074

3:44PM M03.00025 Cilia-driven mixing and transport in complex geometries, HANLIANG GUO, HAI ZHU, SHRAVAN VEERAPANENI, University of Michigan, Ann Arbor — Cilia and flagella are self-actuated microtubule-based structures that are present on many cell surfaces, ranging from the outer surface of single-cell organisms to the internal epithelial surfaces in larger animals. A novel and exciting research direction is in vitro cell cultures is the development of engineered tissues in microfluidic chips, so called 'organs-on-chips'. A fast and reliable numerical method that simulates the coupled hydrodynamics of cilia and the constituent particles in the fluid such as rigid particles, drops or cells would be useful to not only understand several disease and developmental pathologies due to ciliary dysfunction but also to design microfluidic chips with ciliated cultures for some targeted functionality, e.g., maximizing fluid transport or particle mixing. Here we propose a hybrid numerical scheme that employs the boundary integral method for handling the channel and particle boundaries and the method of regularized Stokeslets for handling the cilia. The algorithm is efficient, easy to implement and scales linearly with the problem size. We provide several examples demonstrating the effects of complex geometries on cilia-generated fluid mixing as well as the cilia-particle hydrodynamics.

3:45PM M03.00026 Cryptographic analysis of chaotic fluid flows¹, WILLIAM GILPIN, Harvard Quantitative Biology Initiative — In computer science, hash functions are elementary operators that convert arbitrary-length inputs into finite-length outputs. We describe a direct analogy between these functions and the motion of particles advected by fluid flows, and we show that, when the governing flow is chaotic, hydrodynamic hash functions exhibit statistical properties typically associated with hash functions used for digital cryptography. These include non-invertibility, sensitivity to initial input, and avoidance of collisions, in which two similar inputs produce the same output. We show that this analogy originates from the tendency of certain chaotic flows to braid together particle trajectories across space in time in an irreducible manner, and we describe how techniques used to probe the properties of digital hash functions may be used to characterize the properties of flows when only limited observational data is available. Our findings have potential applications as microfluidic proof-of-work systems, as well as for characterizing large-scale transport by ocean flows and biological microswimmers.

¹Harvard Quantitative Biology Initiative, NDSEG Fellowship

3:46PM M03.00027 Development of a Microfluidic Device to Sort Sperm based on Rheotaxis Effect, AFROUZ ATAEI, ANDY W.C. LAU, WASEEM ASGHAR, Florida Atlantic University — The first step of in-vitro fertilization is to sort out the motile sperm from the non-motile ones. Currently, centrifugation based sperm swim-up and density gradient separation are commonly used methods. However, these methods reduce sperm quality during the repetitive centrifugation steps and isolate sperm with high DNA fragmentation. In this work, we construct a microfluidic device based on the observation that motile sperm can swim against the flow within a specific range of flow rates. This sperm sorting device consists of two chambers, separated by a filter. After 45 minutes the sorted motile sperm are collected from the top retrieval chamber and is placed on a glass slide for visual inspection and data collection. We find that 1) the most motile and functional sperm pass selectively through the micropores against the flow, 2) the optimum flow rate gives the highest concentration of motile sperm, the lowest DNA fragmentation and higher percentage of morphologically normal sperm compared to stock sample. Taken together, our device provide an efficient way to sort sperm without centrifugation.

3:47PM M03.00028 Bending of charged bilayer membranes¹, HAMMAD FAIZI, CODY REEVES, PETIA VLAHOVSKA, Northwestern University — Cells and internal cellular organelles are enveloped by membranes composed primarily of lipid bilayers. The bilayer bending rigidity (resistance to changes in curvature) plays a crucial role in cell deformations. We explore the effect of transmembrane potential on the bending rigidity. We experimentally analyze the dependence of bending rigidity on bilayer charge out of equilibrium due to charge accumulated near the membrane surfaces by an applied electric field. We measure the membrane bending modulus with three different techniques using the same giant unilamellar vesicle: equilibrium shape fluctuations, electrodeformation based on a frequency sweep, and shape fluctuations in the presence of uniform AC field.

¹funded by NSF-CMMI awards 1748049 and 1740011

3:48PM M03.00029 ABSTRACT WITHDRAWN —
Yasuda, Nonlinearity — Pattern formation of chemotactic bacteria is investigated numerically and theoretically based on a Boltzmann-like kinetic transport equation for chemotactic bacteria, say a kinetic model. The MC results illustrate the pattern formation mechanism of chemotactic bacteria induced by the stiff chemotactic response. Furthermore, the unstable modes are always bounded as is observed in Turing instability, so that instabilities are never growing. Our study reveals that slipper shape is the preferred equilibrium shape in low confinement (n = 0) while a transition to parachute shape takes place as n increases. Outward migration tendency due to higher viscosity contrast causes the shape transitions to occur at higher C_0 when λ = 0 compared to the case when λ = 1, resulting in a phase diagram shifted towards higher C_0.

1Supported by NSFC (11832017, 11572334)

3:51PM M03.00032 ABSTRACT WITHDRAWN —

3:52PM M03.00033 Numerical analysis of the pattern formation of chemotactic bacteria based on a kinetic transport model — SHUGO YASUDA, University of Hyogo — Pattern formation of chemotactic bacteria is investigated numerically and theoretically based on a Boltzmann-like kinetic transport equation for chemotactic bacteria, say a kinetic chemotaxis model. In the theoretical part, we discover a novel instability mechanism, i.e., stiff-response-induced instability [B. Perthame& S. Yasuda, Nonlinearity 31 (2018) 4065], where the uniform state of the bacterial population becomes unstable when the stiffness of the chemotactic response of bacteria is sufficiently large. Furthermore, the unstable modes are always bounded as is observed in Turing instability, so that the pattern formation occurs. A Monte Carlo method is also developed based on the kinetic chemotaxis model [S. Yasuda, J. Comput. Phys. 330 (2017) 1022] and the MC method is applied to the pattern formation problems of chemotactic bacteria in one- and two-dimensional spaces. The MC results illustrate the pattern formation mechanism of chemotactic bacteria induced by the stiff chemotactic response.

1Japan Society for the Promotion of Science under the KAKENHI grant No 17H01083

3:53PM M03.00034 3D vesicle microcirculation — DHVANIT AGARWAL, GEORGE BIROS, Oden Institute for Computational Engineering and Sciences, University of Texas at Austin — We study numerically the problem of equilibrium shapes of three-dimensional vesicles in confined axisymmetric Poiseuille flow. We explore a range of the following relevant dimensionless parameters: 1) reduced volume (a), defined as the ratio of volume of vesicle to the sphere of same area as vesicle, 2) viscosity contrast, λ, defined as the ratio of viscosities of internal and external fluids, 3) confinement ratio (C_n), defined as the ratio of vesicle diameter over channel width, 4) capillary number (C_0), measuring flow strength over membrane bending rigidity. We present a phase diagram of equilibrium shapes of vesicles, both with and without viscosity contrast. Our study reveals that the preferred equilibrium shape in low confinement (C_n < 0.5) while a transition to parachute shape takes place as C_n is increased. For high confinement (C_n >> 0.5), the force exerted by the confining walls dominates, causing the vesicle to mostly take axisymmetric shapes. Outward migration tendency due to higher viscosity contrast causes the shape transitions to occur at higher C_0 when λ = 0 compared to the case when λ = 1, resulting in a phase diagram shifted towards higher C_0.

3:54PM M03.00035 Design and validation of a microfluidic pillar device to study hemostasis under flow — HARI HARA SUDHAN LAKSHMANAN, Oregon Health and Science University, ADITYA PORE, Texas Tech University, RACHEL THOMPSON, Oregon Health & Science University, University of Connecticut, JEEVAN MADDALE, West Virginia University, PATRICK JURYNE, JOSEPH SHATZEL, Oregon Health & Science University, SIVA VANAPALLI, Texas Tech University, OWEN MCCARTY, Oregon Health & Science University — Hemostasis is an active process between plasma and blood cells, resulting in thrombin generation, platelet activation and fibrin formation to generate a hemostatic plug that staunches blood loss following vessel injury. The events that support hemostasis (outside the blood vessel) versus thrombosis (inside the blood vessel) are distinct in part due to the rheology of blood flow that differentially distributes blood constituents inside and outside blood vessels. We created an in vitro ‘bleeding chip’ to study the spatial dynamics and cell biology of hemostasis under shear flow. The bleeding chip consists of two orthogonal channels, with series of 3 pillars spaced 10 microns apart at the intersection of the channels acts as a model of endothelial cell barrier function between the intravascular and extravascular space. The bleeding channel is coated with extracellular matrix proteins. We found that platelets aggregate at or behind the pillars as a function of shear rate. Activation of the coagulation cascade staunches blood flow in the bleeding channel while blocking platelet function or coagulation prevented formation of a hemostatic plug. Based on the percolation theory of fluid dynamics, we will discuss the impact of platelet interactions during hemostasis.
to each of the phases and the aspect ratio of the particles are varied. Layers arresting coalescence with reduction of tumbling effects in the arrested state. The effects are changed when the affinity of the particles transition and droplet coalescence process is drastically reduced in presence of ellipsoids of varied hydrophobicity leading to formation of particle is observed with droplet deformation, coalescence and eventual collapse with the emulsion start breaking at the top of the cell. However, this effect govern the key features of elliptical particle-laden flows. As the denser liquid phase drains out, the dense to sparse emulsion transition with the orientation of the particles need to be accounted for because the mutual competition between these phenomena coupled with tumbling Model with long-range repulsive and frustrated short range attractive interactions. Both fluid-particle and particle-particle interactions along elliptical particles at droplet interfaces in a gravity-driven draining emulsion system consisting of a Newtonian fluid using 2D Lattice Boltzmann requires 60.6% less pressure drop than the reference. The results indicate the high potential of using charged nanoporous in reverse osmosis water desalination performance.

For the negatively charged membranes, optimum rejection efficiencies of 94% and 93% are obtained for Na\(^+\), \(\text{Cl}^-\) ions respectively, with 35% lower pressure drop than the reference. Simulations are performed for hydraulic pore diameter membrane as large as 14.40 Å with four different electric charges distributed on the pore surfaces. Pressure-driven flows are induced by moving specular reflection boundaries with a constant speed. Molecular dynamics (MD) simulations. Pressure-driven flows are induced by moving specular reflection boundaries with a constant speed. Dissipative relaxation of the order parameter at the boundary. This relaxation towards contact angle equilibrium involves a contact line friction \(\mu_f\). We thus compare interface shapes obtained from a phase-field diffusive-interface model with the results of molecular dynamics (MD) simulation using the GROMACS code. The setup is a simple Couette flow between two plates, with a vapor droplet sheared in the middle of the domain. Another comparison is performed between MD and the sharp interface model with a slip length and a Generalized Navier Boundary condition. The sharp interface model is implemented using a VOF method. The role of diffusion across the interface, which is possible in the diffuse interface model, is of particular interest.

Positively and negatively charged single-layer nanoporous graphene membranes are investigated for applications in water desalination using molecular dynamics (MD) simulations. Pressure-driven flows are induced by moving specular reflection boundaries with a constant speed. Simulations are performed for hydraulic pore diameter membrane as large as 14.40 Å with four different electric charges distributed on the pore edges. Salt rejection efficiencies and the resulting pressure drops are compared with the obtained free-living case of 9.9 Å diameter uncharged nanoporous graphene membrane, which exhibits 100% salt rejection with 35.02 MPa pressure drop at the same flow rate. For the positively charged membranes, \(q = 9e\) shows 100% and 98% rejection for Na\(^+\) and Cl\(^-\) ions respectively, with 35% lower pressure drop than the reference. For the negatively charged membranes, optimum rejection efficiencies of 94% and 93% are obtained for Na\(^+\) and Cl\(^-\) ions with \(q = -6e\), which requires 60.6% less pressure drop than the reference. The results indicate the high potential of using charged nanoporous in reverse osmosis (RO) desalination systems with significantly enhanced performance.

We study the dynamics of randomly placed two-dimensional elliptical particles at droplet interfaces in a gravity-driven draining emulsion system consisting of a Newtonian fluid using 2D Lattice Boltzmann Model with long-range repulsive and frustrated short range attractive interactions. Both fluid-particle and particle-particle interactions along with the orientation of the particles need to be accounted for because the mutual competition between these phenomena coupled with tumbling effect govern the key features of elliptical particle-laden flows. As the denser liquid phase drains out, the dense to sparse emulsion transition is observed with droplet deformation, coalescence and eventual collapse with the emulsion start breaking at the top of the cell. However, this transition and droplet coalescence process is drastically reduced in presence of ellipsoids of varied hydrophobicity leading to formation of particle layers arresting coalescence with reduction of tumbling effects in the arrested state. The effects are changed when the affinity of the particles to each of the phases and the aspect ratio of the particles are varied.

\(^1\) NASA Grant NNX13AQ22G

\(^1\) A. Capetillo was funded by Conacyt
4:00PM M03.00041 Confinement effects on natural gas within carbon nanotubes. ALEXANDRO KIRCH, TERESA LANNNA, NAIYER RAZMARHA, JULIO MENEGHINI, CAETANO MIRANDA, Universidade de So Paulo — The current membrane technology for gas separation displays a low flow rate and/or low selectivity. Improvements in the separation technologies are desirable for natural gas usage as a fuel supply under carbon restraints. Advances in the membrane functionality may be favored by the development of nanotechnology. Notably with the emergence of the nanostructured carbon-based materials could lead to significant phase separation devices due to their fast mass flow and regular pore size. However, the separation process depends on fine-tuned properties of the pore structure which nowadays could be benefited by atomistic level research studies using molecular dynamics simulations (MD). In particular, the understanding of the fluids basic properties in confined geometries and interfaces plays a central role in the separation process optimization. In this work, we investigated the structure and transport of natural gas within carbon nanotubes in the light of fully atomistic MD simulations. Firstly, molecular models of natural gas are explored and confinement and interface effects on gas selectivity were analyzed with decreasing nanotube diameter. Finally, we could evaluate the underlying molecular mechanisms which could lead to the phase separation and influencing nanoflows.

1The authors acknowledge the sponsorship of the Research Centre for Gas Innovation (FAPESP Proc. 2014/50.279-4) and SHELL Brasil.

4:01PM M03.00042 ABSTRACT WITHDRAWN —

4:02PM M03.00043 Effect of Volume Fraction on Droplet Break-up in an Emulsion flowing through a Microfluidic Constriction. ALISON BICK, SINDY K. Y. TANG, Stanford University — We report the effect of droplet volume fraction on the break-up of droplets within an emulsion flowing as a two-dimensional monolayer through a tapered microchannel into a constriction. A concentrated emulsion was injected into the channel, and an additional continuous phase was injected on-chip to dilute the emulsion to achieve different effective volume fractions. At a fixed flow rate, the break-up fraction decreases significantly when the droplet volume fraction \( \phi \) decreases below \( 0.50 \). This result is consistent with our previous report that droplet break-up arises primarily from droplet-droplet interactions. Furthermore, an optimal location for the introduction of the additional continuous phase to dilute the emulsion was identified as approximately equal to one to two droplet diameters upstream of the constriction. Away from this optimal location, the dilution of the emulsion is ineffective. Assuming a tolerance of a maximum break-up fraction of \( 0.1 \), diluting the emulsion 2.1 times from \( \phi = 0.85 \) to \( \phi = 0.40 \) increases the throughput by \( \sim 1.3 \) times. Consequently, although a higher emulsion volume fraction packs more drops per unit volume, the propensity of the drops to undergo break-up limits the throughput of the process if droplet integrity and assay accuracy are to be maintained.

4:03PM M03.00044 Fine-scale dynamics of bubble induced turbulence with and without breakup and coalescence. IMMANUVEL PAUL, Stanford University, BRUNO FRAGA, University of Birmingham, MICHAEL DODD, Stanford University, CHRIS LAI, Las Alamos National Laboratory — We study bubble induced turbulence (BIT) through a swarm of air bubbles rising in a vertical column. We consider two types of bubbles: one without and another with breakup and coalescence. The immersed boundary method is used to model the bubbles without breakup and coalescence while the volume of fluid is utilized for the other case. The void fraction of air bubbles is 0.5% at the start in both the simulations. We focus on the quasi-steady stage of the temporal evolution of the BIT. For this stage, we also note the -3 slope in the energy spectrum as reported in the experiments. We further explore the fine-scale turbulence dynamics by studying the evolution equation of the velocity gradient tensor (VGT) in Q-R space. We observe an entirely different conditional mean trajectories (CMT) of R and Q in both the BIT simulations. Here, the trajectories no longer spiral towards the origin as in the case of the homogeneous isotropic turbulence. Rather, the CMTs evolve to a particular line on the right branch of D=0 line leading to divergence in finite time. This is accompanied with the larger magnitude of the pressure Hessian term compared with all other terms in the VGT evolution equation.

4:04PM M03.00045 Secondary flows and bubbly drag reduction on heterogeneously rough surfaces in Taylor–Couette turbulence. PIM BULLEE, DENNIS BAKHUIS, RODRIGO EZETA, SANDER HUISMAN, Twente University, CHAO SUN, Tsinghua University, ROB LAMMERTINK, DETLEF LOHSE, Twente University — We investigate turbulent flow at Reynolds numbers (Re) ranging from \( 0.5 \times 10^5 \) till \( 1.8 \times 10^6 \) over surfaces that are composed of rough and smooth patches. Our main interest is in how the structure of the flow is influenced by this alternating rough and smooth surface, and how this changes the position of air bubbles in the flow. For this we use the Taylor–Couette geometry, which allows for accurate drag and flow measurements, as well as flow visualisations through its transparent outer cylinder. The inner cylinder wall is covered with sandpaper, leaving room between bands to form a constant pattern in axial direction. This results in a strong secondary flow pattern of radially outward flow on the rough patches and radially inward flow on the smooth patches. Surprisingly, air bubbles do not follow this secondary flow, but rather accumulate on top of the rough wall sections in the flow. Here, locally, the drag is largest and so the drag reducing effect of the bubbles is felt strongest. Therefore, a larger maximum value of bubbly drag reduction is found for the alternating rough and smooth inner cylinder walls compared to the completely rough and completely smooth inner cylinder walls.

1NWO-TTW (project 14504)

4:05PM M03.00046 Investigation of Cavitation Regimes Using Large Eddy Simulations. MRUGANK BHATT, KRISHNAN MAHESH, University of Minnesota — Cavitation occurs over different regimes, ranging from inception to massive regions of vapor. We use LES to examine the different regimes of cavitation for i) partial cavitation over a wedge and ii) cavitation over a marine propeller. Cavitation over a wedge is simulated at \( Re = 200,000 \) and \( b=2.47, 1.89 \) and 1.78 demonstrating respectively the incipient, the transitory and the periodic regimes. Cavitation over five bladed marine propeller (P4381) is studied at \( Re = 894,000 \) and \( = 0.40 \) increases the throughput by \( \sim 1.3 \) times. Consequently, although a higher emulsion volume fraction packs more drops per unit volume, the propensity of the drops to undergo break-up limits the throughput of the process if droplet integrity and assay accuracy are to be maintained. This result is consistent with our previous report that droplet break-up arises primarily from droplet-droplet interactions. Furthermore, an optimal location for the introduction of the additional continuous phase to dilute the emulsion was identified as approximately equal to one to two droplet diameters upstream of the constriction. Away from this optimal location, the dilution of the emulsion is ineffective. Assuming a tolerance of a maximum break-up fraction of \( 0.1 \), diluting the emulsion 2.1 times from \( \phi = 0.85 \) to \( \phi = 0.40 \) increases the throughput by \( \sim 1.3 \) times. Consequently, although a higher emulsion volume fraction packs more drops per unit volume, the propensity of the drops to undergo break-up limits the throughput of the process if droplet integrity and assay accuracy are to be maintained.

1This work is supported by the Office of Naval Research.
4:06PM M03.00047 Dynamics of gas bubbles and slugs in 2-dimensional porous media, PETR DENISSENKO, University of Warwick, RUFAT ABIEV, St. Petersburg State Institute of Technology, SAM TUCKER HARVEY, EVGENY REBROV, University of Warwick — A 3d-printed 2-dimensional channel is used to study 2-phase flow in a bench-scale trickle-bed reactor. As the maximum size of a spherical bubble is limited by the pore size, slugs with complex shapes form when bubbles merge. The slugs do not move constantly but perform intermittent manoeuvres. Break-up, coalescence, and propagation dynamics of the slugs are studied in relation to their length. The balance between pressure, viscosity, and surface tension terms is considered to predict slug propagation dynamics. Interaction between slugs is considered. Results are interpreted in the context of the gas hold-up and the flow quality in relation to mass transfer between gas phase and liquid phase.

4:07PM M03.00048 Application-Specific Microfluidic Networks, FAIZ K. MOHAMMAD, VEDULA MURALI, RAGHUNATHAN RENGASWAMY, None, SENAI TEAM — Microfluidic devices have applications in various fields. This paper reports a simple approach for designing application-specific microfluidic networks with low effort and time. Here, we also demonstrate how this approach can be utilized to find networks that can perform logic computations based on droplets motion inside the network. We applied our approach to find networks that can control the motion of drops for logical computations, i.e., AND (·) gate where a drop will emerge from an arm only if drops enter the network simultaneously, and OR (+) gate when a drop will emerge from an arm whenever drops enter the network. Finding a device which can do AND (·), OR (+) logic computations might be easy (has been demonstrated) and can be achieved by trial and error. However, for more complex logical operations, trial and error approaches will be time consuming and unreliable. However, the proposed approach can handle these problems quite easily. This approach can also work with multiple drops entering from different source positions into the network to be used in more complex logical operations. We will describe computationally identified device designs for such logic computations in this work.

4:08PM M03.00049 Acoustic droplet vaporization on hydrophobic and hydrophilic solid surfaces, SEHO KWON, GIHUN SON, Sogang University — Acoustic-driven vaporization of volatile droplets, such as dodecafluoropentane (DDFP) droplets with a lower boiling temperature of 29°C, receives attention as a promising means for medical diagnostic and therapeutic applications. The acoustic droplet vaporization (ADV) process on hydrophobic and hydrophilic surfaces is experimentally investigated using a microscope and high-speed camera system with temporal and spatial resolutions of 150,000 frames/s and 5 pixels/um, respectively. A water-repellent agent is coated on a slide glass for a hydrophobic surface whereas an anti-fogging agent for a hydrophilic surface. The DDFP droplet has a contact angle of 105 degree on the hydrophobic surface and 45 degree on the hydrophilic surface. Ultrasound with a center frequency of 5 MHz was applied to DDFP droplets immersed in liquid water. The present ADV experiments show that the droplet vaporization rate is higher on the hydrophilic surface than on the hydrophobic surface. The droplet-bubble compound is always attached on the hydrophobic surface during the ADV process, but the compound grown on the hydrophilic surface slides off while leaving a small droplet portion. The effect of ambient water temperature on the droplet and bubble motion was quantified.

4:09PM M03.00050 Closed loop control of the acoustic energy shielding by cavitation bubble clouds1, KAZUKI MAEDA, Stanford University, ADAM MAXWELL, University of Washington — A closed loop control system is developed to regulate the transmission of ultrasound burst into a solid obstacle that is shielded by a layer of cavitation bubble clouds. The study is motivated by interest in improving the efficacy of kidney stone comminution during a recently proposed ultrasound-based lithotripsy. In the system, pulses of ultrasound with a frequency of $O(100)$ kHz and an amplitude of $O(1)$ MPa are focused on a stone model from a multi-element array transducer with a pulse-repetition-frequency (PRF) of $O(10)$ Hz. The far-field, bubble-scattered acoustic waves are concurrently measured at the transducer arrays. With a high PRF, the layer of bubbles is excited on the proximal side of the stone and scatters a large portion of the incoming acoustic energy. A data-driven, reduced-order model is used to estimate the portion of the energy transmitted into the stone from the acoustic measurement in real-time. Based on the offset of the estimation from a set point, a proportional-integral controller modulates the PRF to control cavitation. The controller showed favorable performance during $O(100)$ s of continuous insonification. Lastly, the system is used to identify the optimal set point that maximizes the effective rate of energy transmission into the stone.  

1Funding supported by NIH P01-DK043881

4:10PM M03.00051 Ensemble-based Data Assimilation Methods for Viscoelastic Material Rheometry during Bubble Collapse1, J. JEAN-SEBASTIEN SPRATT, MAURO RODRIGUEZ, KEVIN SCHMID-MAYER, TIM COLONIUS, California Institute of Technology — We examine ensemble-based data assimilation methods for viscoelastic material rheometry, where observation of the radius versus time of a collapsing spherical cavitation bubble is used, together with a physical model for the bubble dynamics, to infer the surrounding material’s mechanical properties. Such ensemble-based stochastic methods are attractive for this type of problem, as they fully capture the nonlinear dynamics while keeping computational costs low given the large number of state variables. The ensemble Kalman filter (EnKF), iterative ensemble Kalman smoother (IEnKS), and a hybrid ensemble-based 4D-Var (En4D-Var) method are compared. These are first validated against simulated data with known parameters, and then applied to experimental measurements in water and Polyacrylamide gel [Estrada et. al. 2018, J Mech Phys Solids 112]. We show that the IEnKS and En4D-Var improve on the results of the EnKF as expected for this problem, and outperform existing viscoelastic material characterization methods by achieving comparable or better estimation with reduced computational cost. Each method demonstrates particular advantages, the IEnKS being less expensive and better suited for higher frequency data, and the En4D-Var more robust for sparse data sets spanning longer time.

1This research was supported by a grant from the National Institutes of Health (NIH grant no. 2P01-DK043881) and from the Office of Naval Research (ONR grant no. N0014-18-1-2625)

4:11PM M03.00052 Selective Viscous Withdrawal: Entrainment of Micro-jets and Micro-drops, ZEHAO PAN, JANINE NUNES, NIKI ABBASI, HOWARD STONE, Princeton University, STONE GROUP TEAM — The study of microscale drop or jet generation has catalyzed the invention of numerous biomedical and industrial technologies. Existing shear-based technologies require active control of both the continuous and the dispersed phases, therefore limiting their capacity for large scale parallelization. A selective viscous withdrawal system is comprised of a nozzle immersed in one fluid near a fluid-fluid interface, where the application of a withdrawal flow can cause the entrainment of the second fluid into the nozzle forming jets or droplets. Since no active or independent flow rate control of the two fluids is needed, this system promises greater capacity for scale up of droplet or jet production. Here, we experimentally study the formation of jets and droplets in a selective viscous withdrawal system, and perform a scaling analysis of the jet and drop sizes with respect to key operation parameters including the flow rate, nozzle-interface distance, nozzle diameter, viscosity, and interfacial tension. Our work lays the groundwork for further application-driven explorations.
4:12PM M03.00055 Touch Down of a Sphere in Viscoelastic Media
THEODORE BRZINSKI, Haverford College, PAULO ARRATIA, University of Pennsylvania — Many fluids of practical interest contain polymers and show non-Newtonian behavior such as shear-thinning viscosity and elasticity. Lubrication forces in such fluids depend on conformation changes of polymer molecules and at sufficiently high strain-rates can become nonlinear. Here, we experimentally investigate the (lubrication) forces that arise as a sphere approaches the solid surface as a function of fluid elasticity. We use an Instron with a sphere-and-rod attachment to directly measure the forces experienced by a sphere that approaches a smooth surface in fluids with different levels of elasticity.

4:13PM M03.00054 Prediction of Rheological Behaviour of Unentangled Polymer Solutions in Steady Shear Flows Using a Blob-theory Based Constitutive Model
PRABHAKAR RANGANATHAN, Monash University — Understanding the effects of polymer concentration and flow-induced stretching on hydrodynamic interaction between segments of polymer molecules is the key to accurately predicting the dynamical behavior of unentangled polymer solutions. Conventional wisdom avers that a polymer solution is dilute when its concentration \( c \) is less the concentration \( c^* \) at which isotropic polymer coils begin to overlap and interpenetrate. This picture is simplistic. Inter-chain interaction becomes increasingly important as chains stretch in flow. In semi-dilute unentangled solutions, on the other hand, concentration-dependence is strong at equilibrium, but inter-chain interactions weaken as chains stretch during flow. Thus, dilute solutions self-concentrate, while semi-dilute solutions self-dilute. A constitutive model based on blob concepts is used to examine the concentration dependence of shear-thinning-thickening-thinning in steady shear flows. It is shown that qualitative nonlinear rheological behaviour in strong flows is surprisingly sensitive to the friction coefficient of the Kuhn segment.

4:14PM M03.00055 Rheological properties of encaustic painting
JORGE ARROYAVE, SANDRA ZETINA, Universidad Nacional Autonoma de Mexico, ROBERTO ZENIT, Brown University — Encaustic is a painting technique characterized by the use of wax as the main binding media. In addition to wax and pigments, some resins and solvents are used to create the painting media. Since the material is semi-solid at ambient conditions, the painting is often conducted with the aid of heated pallets and torches. As a result, mastering this technique is extremely challenging. In this work we investigate how the composition of the encaustic affects its rheological properties focusing on the effect of pigment color and concentration. We prepare encaustic paints following Diego Rivera’s, used for his first large-scale mural which was painted with this technique. In addition to determining the viscosity, using a rheometer, we also conducted controlled painting experiments: a device was designed and built to apply a drop of paint to heated surface at different shear rates. We found that all the paints were shear-thinning with small viscoelastic effects; surprisingly, the viscosity of the paints was significantly affected by the pigment color despite the fact that its proportion in the paints was always small (less than 10%). The size of the traces was found to scale with the shear Reynolds number. A discussion on the implications of these results for artists is presented.

4:15PM M03.00056 A Mesh Refinement framework for the Lattice Green’s Function method
KE YU, BENEDIKT DORSCHNER, MARCUS LEE, TIM COLONIUS, California Institute of Technology — We report on progress in developing the lattice Greens function (LGF) technique for solving viscous, incompressible flows on unbounded domains. This method exploits the regularity of the finite-volume scheme on a formally unbounded Cartesian mesh to yield robust (conservative, stable) and computationally efficient solutions. It is spatially adaptive, but of fixed resolution. Here we develop an adaptive mesh refinement strategy compatible with the LGF algorithm. The solution to the pressure Poisson equation is approximated using the LGF technique on a composite mesh where different regions can have different resolutions. This is further combined with an integrating factor technique for the viscous term and an appropriate Runge-Kutta scheme for the resulting differential-algebraic equations. The algorithm is verified and validated with numerical simulations of vortex rings.


3:20PM M04.00001 Local analysis of the clustering, velocities and accelerations of particles settling in turbulence
MOHAMMADREZA MOMENIFAR1, ANDREW D. BRAGG2, Duke University — We use Direct Numerical Simulations (DNS) and 3D Voronoi tessellation to analyze the local dynamics of small inertial particles in isotropic turbulence, considering the effect of Taylor Reynolds number (\( R_e \)), Froude number (\( Fr \)), and Stokes number (\( St \)). In line with previous results using global measures of particle clustering, we find that for small Voronoi volumes, the behavior is strongly dependent upon \( St \) and \( Fr \), but only weakly dependent upon \( R_e \), unless \( St > 1 \). However, larger Voronoi volumes (void regions) exhibit a much stronger dependence on \( R_e \), even when \( St < 1 \). This, rather than the behavior at small volumes, is the cause of the sensitivity of the standard deviation of Voronoi volumes \( \lambda \) that has been previously reported. Particle acceleration results indicate a non-trivial effect of gravity, while results for the fluid acceleration at the particle position call into question the sweep-stick mechanism for clustering. Comparing the local dynamics of particles in clusters to all particles in the flow reveals that while their kinetic energies are nearly the same, the clustered particles settle much faster on average, and this difference grows significantly with increasing \( R_e \).

1Department of Civil and Environmental Engineering, Duke University, Durham, North Carolina 27708, USA
2Department of Civil and Environmental Engineering, Duke University, Durham, North Carolina 27708, USA

3:21PM M04.00002 ABSTRACT WITHDRAWN —

3:22PM M04.00003 Feedback control of a combined two-fluid/electro-spray coaxial injector via real-time measurements and Principal Component Analysis
RODRIGO OSUNA-OROZCO, NATHANIEL MACHICOANE, PETER HUCK, ALBERTO ALISEDA, University of Washington — We present an experimental study of the physics of gas-assisted atomization combined with electro-spray. We leverage a low dimensional representation of the spray, from light attenuation measurements, to implement feedback control in real-time. The laminar liquid stream is injected through a long straight metallic needle at the center of the turbulent gas jet. The liquid is electrically charged by a very strong electric field at the nozzle exit. We apply an external electric field along the spray development region, characterizing the ability of an external sideways force on the individual droplets to modify the structure of the spray in the midfield. We characterize the break-up dynamics using high-speed visualizations in the near field and the droplet sizes and velocities in the midfield using light interferometry. In the implementation of real-time control, we use optical attenuation of light traversing the spray downstream of the nozzle in the mid-field. Low dimensional representations of the radial liquid volume fraction profiles allow for real-time control of the spray based on actuation on the gas total and angular momentum as well as on the external electric field.
3:24PM M04.00005 On The Relationship Between Internal Flow and Jet Dynamics in a Charge Injection Atomizer, WILLIAM DOAK, PAUL CHIAROT, State University of New York at Binghamton, MICROFLUIDICS AND MULTIPHASE FLOW LABORATORY TEAM — Atomization of a dielectric micro-jet is achieved using an electrohydrodynamic (EHD) charge injection process. The atomizer is comprised of a grounded nozzle plate and an internal high voltage probe, with electric potentials up to 20 kV, concentric with the emitting orifice. Dielectric liquid flows through the cavity between the electrodes, impeding electron transport from the probe to ground and imparting charge to the fluid. When the jet is uncharged, it breaks up via an axisymmetric (Rayleigh-Plateau) instability. Once charged, a non-axisymmetric (bending) instability, in addition to the axisymmetric instability, is observed in the jet. Both instabilities grow with increasing jet charge density: the abrupt jet length shortens and the bending amplitude increases. We have found that changes in the jet instability modes are related to the EHD flow induced inside the nozzle. A transparent nozzle was built and the flow was seeded with micro-spheres to study this phenomenon. High-speed microscopy and digital image correlation was used to measure the EHD flow at different jet stability conditions. We found that transitions in the internal flow are associated with changes in the jet stability regimes.

This research was supported by the American Chemical Society Petroleum Research Fund.

This work was supported by the Natural Environment Research Council [grant number NE/R00014X/1].

3:26PM M04.00007 Hindered coalescence and break-up with insoluble surfactants, CAROLINA VANNOZZI1. None — For Capillary numbers (Ca) greater than 0.05, the thin film between two drops undergoing a flow-induced head-on collision, in the creeping flow regime, thins to a steady state thickness hss, as shown in [1]. Here, we analyze numerically this phenomenon in the presence of surfactants. For trace amount of surfactants, hss diminishes with decreasing surfactant interfacial diffusivity (Ds) and , unlike Nemer et al.’s analysis for non-diffusing surfactants[1], surfactants are still present in the dome region. For higher surfactant concentration, hss is present only for very high Ds, while, as Ds decreases, the film drains continuously. As Ca increases the two drops break-up into 4 distinct drops. Interestingly, for low surfactant concentrations, the dome region will thin, becoming a neck. This will continuously thin and stretch as the two extremities depart from each other following the streamlines, while keeping hss constant. This process will proceed until instabilities arise in the neck. Whereas, for high surfactant concentrations, the dome region will thin only at the center and the extremities will keep draining in place. [1] Nemer et al. 2004 Phys. Rev. Lett. 92.

This work was undertaken while I was affiliated with UCSB.

3:28PM M04.00009 Particle aggregates via droplet evaporation on superhydrophobic fractal-like substrates, CAROLA SEYFERT, EVA KROLIS, Physics of Fluids, University of Twente, ERWIN J.W. BERENSCHOT, ARTURO SUSARREY-ARCE, NIELS TAS, Mesoscale Chemical Systems, University of Twente, ALVARO MARIN, Physics of Fluids, University of Twente — Sessile droplets on superhydrophobic substrates are common in nature and technology. In the case of droplets containing solid particles, the evaporation of the solvent turns into an effective tool to aggregate any non-volatile content. The high contact angles and unpinned contact lines of the droplets, induced by hydrophobicity, can lead to a complete recovery of the solid solute in form of aggregates. Under the right conditions, the solid remainder takes the form of highly compact and spherically shaped aggregates, featuring a minimal contact area with the supporting substrate. We investigate the evaporation of colloidal droplets on a new kind of superhydrophobic, micro-structured substrate, featuring fractal-like glass pillars. Such substrates present an intricate geometry with non-flat top surfaces of the pillars. Different sizes and concentrations of monodisperse polystyrene particles lead to various shapes of particle aggregates after the evaporation of the solvent.

We would like to acknowledge the financial support of the ERC-StG 678573 Nanopacks.

3:29PM M04.00010 Colloidal crystallization in cylindrical geometry: Effect of particle wettability on banding, TEJASWII SOORI, YING SUN, Drexel University — A colloidal drop constrained within a capillary tube subjected to evaporation results in crystallization and particle banding. Past studies have shown the width and spacing of bands to increase with concentration for cylindrical geometries. The radius of curvature \( R \) and capillary length \( \ell_c \), for drops over planar substrates are usually of the same order of magnitude, while for capillary tubes the radius of curvature \( R \) is equal to the capillary radius \( r \) which can be much smaller than the capillary length \( \ell_c \). Taking advantage of this fact, we use colloidal drops containing particles with different wetting properties in capillary tubes of radii \( r = 400 \) and \( 750 \) \( \mu \)m at initial concentrations \( \phi = 0.1 \) and \( 0.5 \) % wt. We perform all evaporation experiments at isothermal, controlled humidity conditions and use an in-house MATLAB code to analyze the images captured via a CCD camera to measure the transient quantities like contact angle and contact line position. In this talk, we report the results quantifying the effect of particle wettability on deposition dynamics, contact line motion, and banding.
3:30PM M04.00011 Determining time scales for directed assembly of particles by shear flow and electric field1. MINAMI YODA, ANDREW YEE, Georgia Institute of Technology, HAJIME ONUKI, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology — Suspended colloidal polystyrene particles assemble into structures called “bands” when subject to shear flow and a dc electric field. These bands exist only within a few μm of the wall, and have been observed over a wide range of conditions in combined Poiseuille and electroosmotic “counterflow” through microchannels, even at particle volume fractions as low as 0.33 ppm. There appear to be three stages in this process: 1) Accumulation, where particles are concentrated near the channel wall; 2) Band formation, with a relatively large number of unstable bands; and 3) Stable bands. The time scales for these stages were determined from evanescent-wave visualization images acquired at different streamwise channel locations. The standard deviation in the image grayscales was used to estimate the time required to reach the band formation stage, and compared with a similar time scale based instead on the time when the first band is observed. The mean grayscale, which was used instead to estimate the time to reach the stable bands stage, appears to have an exponential growth during the accumulation stage, and reaches its maximum value during the band formation stage, before decreasing to a relatively constant value in the stable bands stage.

1Supported by US Army Research Office and Japan Society for the Promotion of Science

3:31PM M04.00012 Defining dimensionless parameters for electrohydrodynamic field-directed nanowire assembly.1, RUSTOM BHILADVALA, University of Victoria, MAHSID SAM, QuirkLogic Inc. — We present an approach for defining dimensionless parameters based on competing forces on nanoparticles in an electrohydrodynamic assembly process. These forces can either direct or disrupt the assembly process in different assembly situations. We define dimensionless parameters to maximize the ratio of directive to disruptive forces. This work is motivated by promising scientific capabilities that have been demonstrated using nanostructured devices at laboratory scale. Their translation to useful devices over centimeter to meter squared area at reasonable cost, can be facilitated by methods of field-directed assembly of nanostructures. Capillary and viscous forces, dielectrophoresis and dipole-dipole interaction, electrode polarization and electroosmosis, are involved during assembly. These depend on values of physical properties of the nanoparticles and the suspension fluid, design of the electrode pattern and the potential and frequency chosen for the electric field. The value of some of these physical parameters can each simultaneously affect forces that direct and disrupt nanostructure assembly. The tedious trial and error involved in parameter selection is an impediment to nanomanufacturing. A systematic approach to eliminate it is the motivation for this work.

1Support from funding agencies NSERC, MITACS.

3:32PM M04.00013 ABSTRACT WITHDRAWN —

3:33PM M04.00014 Electrorotational instabilities of a drop in a uniform DC electric field1, PETIA VLAHOVSKA, JEREMY KOCH, MICHAEL MIKSIS, Northwestern University — In a uniform electric field, a weakly conducting drop bearing zero net charge initially adopts a prolate or olate spheroidal shape, with both the shape and flow axisymmetrically aligned with the applied field - a classical result from G.I. Taylor. At higher field strengths we find two symmetry-breaking instabilities: a high viscosity drop undergoes Quincke rotation (the global flow acquires a rotational component), while low viscosity drops only develop a secondary flow – a series of surface vortices – in a belt along the drop equator. We explore these phenomena experimentally in a silicone oil/caster oil system to map the region of the vortices-belt instability as a function of fluid viscosity and electric field strength.

1funded by NSF-CBET award 1704996

3:34PM M04.00015 Chaotic dynamics of a Quincke rotor in 3D1. GERARDO PRADILLO, HAMID KARANI, PETIA VLAHOVSKA, Northwestern University — The Quincke effect is an electrohydrodynamic instability which gives rise to a torque on a dielectric particle in a uniform DC electric field. The equations which describe the resulting rotation are known to map onto the Lorenz equations (Peters et al, Chaos (2005)), predicting the existence of a second bifurcation upon which rotation is no longer steady. In this presentation we discuss the dynamics of the Quincke rotor in 3D, using the recently discovered hovering state (Pradillo et al, Soft Matter (2019)) at high field strengths. We experimentally confirm the existence of chaotic motion and demonstrate the presence of periodic regimes in this previously unexplored 3D system. Our experimental results are compared to the solutions of the fully three-dimensional model.

1supported by NSF awards CBET- 1704996 and CMMI- 1740011

3:35PM M04.00016 Electrohydrodynamic instability of a suspended liquid film, MO-HAMMADHOSSEIN FIROUZJIA, DAVID SAINTILLAN, University of California San Diego — Electrohydrodynamic instabilities at liquid-liquid interfaces are of high importance due to their relevance in various microfluidic applications. In this work, we analyze the stability of a system of three superposed layers of two immiscible liquids subject to a normal electric field. Following the leaky-dielectric model, the interfaces admit conduction, advection by the flow, and finite charge relaxation. We use this model to perform a linear stability analysis and uncover regimes of instability in terms of the relevant dimensionless groups of the problem.

3:36PM M04.00017 Modulation of the streaming potential and slip characteristics in electrolyte flow over liquid filled surfaces1. BEI FAN, PRABHAKAR BANDARU, University of California San Diego — A significant enhancement in the streaming potential (Vs) was obtained in experiments considering the flow of electrolyte over liquid-filled surfaces (LFSs), where the grooves in patterned substrates are filled with electrolyte immiscible oils. Such LFSs yield larger Vs (by a factor of 1.5) compared to superhydrophobic surfaces, with air-filled grooves, and offer tunability of electrokinetic flow. Through changing the filling oils, it is shown that the density, viscosity, conductivity, surface tension as well as the dielectric constant of the filling oil in the LFS, determine Vs. Considering the concentration polarization phenomenon for nonhomogeneous charged LFS, the hydrodynamic slip length is inversely proportional to the dielectric constant of the filled oil. Relating the hydrodynamic slip length to the obtained Vs offers insight into flow characteristics, as modulated by the liquid-liquid interfaces in the LFS.

1National Science Foundation (NSF: CBET 1606192)
3:37PM M04.00018 Low-voltage onset of electokinetic mixing at heterogeneous charge selective interfaces. ANNE BENNEKER, University of Calgary, BURCU GUMUSCU, ERNEST DERCKX, ROB LAMBERTINK, JEFFERY WOOD, University of Twente — Electokinetic mixing of the concentration boundary layer adjacent to charge selective interfaces can enhance the performance of electrodialysis systems and fuel cells by increasing the transport of ions towards the interface. Traditionally, electokinetic instabilities occur at relatively high potentials, significantly reducing the system efficiency. In this work, we experimentally show that altering the topology of the charge selective interface can induce electokinetic mixing at low potentials using patterned charge selective hydrogels. Fluorescence microscopy for mapping local ion concentration is combined with electrical characterization to unravel the development of ion depletion zones and the onset of electokinetic mixing. For different geometries, we find that the development of these depletion zones is distinctly different as a result of the distribution of the field lines through the different geometries. Enhancement in the total transport is observed with increasing system heterogeneity as a result of electrosomotic contributions to the charge transport, starting a low applied potentials. This indicates that inducing non-uniform electric fields using membrane topology variations and spacers is a promising route for increasing charge transport to the interface.

3:38PM M04.00019 ABSTRACT WITHDRAWN —

3:39PM M04.00020 Contact angle effect on the drop impact onto a solid substrate. ZHEN JIAN1, International Center for Applied Mechanics, School of Aerospace, Xi’an Jiaotong University, Xi’an 710049, P.R.China, CHRISTOPHE JOSSERAND, STEPHANE ZALESKI, PASCAL RAY, Sorbonne Universite, CNRS, UMR 7190, Institut Jean Le Rond d’Alembert, F-75005 Paris, France — As drop impacts onto a solid substrate, abundant outcomes such as splashing, bouncing can be observed under different conditions including liquid, gas and solid properties. We study the contact angle effect on the splashing of drop impact by direct numerical simulation with open source codes Gerris and Basilisk. For a typical partial wetting substrate with a contact angle of 90°, we obtain a phase diagram of splashing or not by varying the density and viscosity ratio between gas and liquid. Splashing can be either created by changing the contact angle to smaller than 90°, or eliminated with a contact angle larger than 90° with all the other properties constant. The wettability plays a role in the formation of splashing during drop impact on solid.

3:40PM M04.00021 An Evaluation of Droplet Breakup Characteristics for Low- and High-Speed Vehicle Impacts. MICHAEL KINZEL, University of Central Florida, JASON TURNER, University of Illinois Urbana Champagne, BRENDON CAVAINOLO, CAROLINE ANDERSON, University of Central Florida, UIUC TEAM, CFAL TEAM — In this work, computational fluid dynamics (CFD) predictions are used to simulate and study the evolution of a droplet in its approach of aerodynamic surfaces. The CFD effort is based on the volume-of-fluid (VOF) method in a formulation that contains the compressibility of both the liquid and gas. Using this formulation, the calculations of the breakdown of the droplet on impingement are directly simulated. These studies include evaporation and interaction with aerodynamic interference. The droplet travel speed depending on the threshold in oscillation acceleration, the water droplet started to travel to the down direction gradually increasing the travel speed. The opposite substrates propelled water droplets of few microliters preferentially in the down-hill direction relative to the sawtooth geometries. Above a threshold in oscillation acceleration, the droplet may either coalesce within less than one droplet diameter distance from the turning point or travel many droplet diameters. We find a clear bi-modal distribution in droplet travel distance at this transition point and present a physical model to explain this interaction.

3:41PM M04.00022 Contact line friction driven droplet transport over asymmetric sawtooth surface microstructures. YAERIM LEE, The University of Tokyo, GUSTAV AMBERG, KTH Royal Institute of Technology, JUNICHIRO SHIOMI, The University of Tokyo, CAROLINE ANDERSON, University of Central Florida, UIUC TEAM, CFAL TEAM — In this work, computational fluid dynamics (CFD) predictions are used to simulate and study the evolution of a droplet in its approach of aerodynamic surfaces. The CFD effort is based on the volume-of-fluid (VOF) method in a formulation that contains the compressibility of both the liquid and gas. Using this formulation, the calculations of the breakdown of the droplet on impingement are directly simulated. These studies include evaporation and interaction with aerodynamic interference. The droplet travel speed depending on the threshold in oscillation acceleration, the water droplet started to travel to the down direction gradually increasing the travel speed. The opposite substrates propelled water droplets of few microliters preferentially in the down-hill direction relative to the sawtooth geometries. Above a threshold in oscillation acceleration, the droplet may either coalesce within less than one droplet diameter distance from the turning point or travel many droplet diameters. We find a clear bi-modal distribution in droplet travel distance at this transition point and present a physical model to explain this interaction.

3:42PM M04.00023 Flow field near Contact Lines : Role of Inertia1, ANJISHNU CHOUDHURY, CHARUL GUPTA, HARISH N. DIXIT, Indian Institute of Technology Hyderabad — The dynamics of contact line involves the movement of two immiscible fluids in contact with a solid surface. An analytical solution for Stokes flow near a moving contact line was solved by Huh & Scriven (JCIS, 1971) which suggests a simple classification of the flow field based on viscosity ratio, $R$ and contact angle, $\theta$. But experiments by Savelski et al. (JCIS, 1995) and Ito et al. (Trans. Vis. Soc. Japan, 2009) find flow fields which differ from predictions of Huh & Scriven theory. Our work is an attempt to resolve this contradiction by exploring the kinematics near the contact line through accurate simulations and scaling exponent on our attention focus our attention on fields in the region of $O(l_c)$, the capillary length. Excellent agreement between simulations and theory is found in the viscous limit. A careful examination of the parameters used in the reported experiments suggests that $Re$ based on $l_c$ is not small, hence inertial effects may not be negligible. We observe a stagnation point on the interface at length $l_i$ for moderate $Re$ and at $O(1)$ We. The obtained flow field potentially resolves the contradiction between theory and experiments. Experiments are underway to validate these flow field and will be discussed during the meeting.

3:43PM M04.00024 Droplet baseball: skirting droplets rebounding from a rigid wall1, JACOB HALE, ALYSSA FISHER, MOMOKA GOTO, ANH LE, MASON LEE, DePauw University — Droplets of 1 cm silicone oil, with an initial velocity component tangent to the surface of a bath of the same fluid, will roll along the surface for hundreds of milliseconds, slowing exponentially until eventual coalescence. A rigid vertical wall, placed in the bath perpendicular to the path of the droplet, forces early coalescence either on the meniscus formed on the wall or after rebounding from the wall. By moving the wall along the line of motion of the droplet, a transition point is found in the droplet’s trajectory in which, at a fixed wall position, the droplet may either coalesce within less than one droplet diameters distance from the turning point or travel many droplet diameters. We find a clear bi-modal distribution in droplet travel distance at this transition point and present a physical model to explain this interaction.

1Douglas A. and Phyllis G. Smith Student Faculty Collaborative Research Fund, Arthur Vining Davis Foundations, DePauw Science Research Fellows, DePauw Faculty Development Program
3:45PM M04.00026 Surface Chemistry Dramatically Influence Cassie-to-Wenzel Transitions in Doubly Reentrant CAVITIES 1, SANKARA NARAYANA MOORTHI ARUNACHALAM, ZAIN AHMAD, RATUL DAS, JAMILYA NAURUZBAYEVA, HIMANSHU MISHRA 2, King Abdullah University of Science and Technology — Surfaces and membranes that can robustly entrap air on immersion in liquids have proven valuable for drag reduction and desalination. Typically, the ability to entrap air is engendered by coating rough surfaces/membranes with water-repellent chemicals. Recently, it has been demonstrated that if microstructures comprising doubly reentrant cavities (DRCs), i.e., cavities that broaden below the inlets and whose walls have T-shaped cross-section, are carved on a surface, can entrap air despite the hydrophilicity of the material. Here, we studied wetting transitions in DRCs with a variety of surface chemistries and compared them with simple cylindrical cavities (SCCs). We realized arrays of DRCs and SCCs on SiO2/Si wafers using photolithography and dry etching processes, and modified surface chemistries of silica to realize water (intrinsically) contact angles, , 0, 40, 112. We found that life-times of Cassie-states could vary from s. The mechanisms underlying the wetting transitions including, the diffusion of the trapped air, the capillary condensation, liquid imbibition, and the release of the trapped air as a bubble will be explained.

1 KAUST Baseline Research Funding BAS/1/1070-01-01
2 Corresponding Author

3:46PM M04.00027 ABSTRACT WITHDRAWN

3:47PM M04.00028 ABSTRACT WITHDRAWN

3:48PM M04.00029 Dynamics of spontaneous emulsification at particle-laden interfaces: a coupling between interfacial microstructures and interfacial flows. , PARISA BAZAZI, HOSSEIN HEJAZI, university of Calgary — Interfaces between two immiscible liquids are omnipresent in nature and industrial processes including enhanced oil production, CO2 sequestration, and drug delivery. A thermodynamically stable dispersion of liquids, i.e. microemulsions, may develop at the fluid interfaces under specific circumstances such as having close to zero interfacial tension. In this work, we investigate the possibility of spontaneous emulsification at the particle-laden oil-water interfaces and examine the extent to which the phenomena are analogous to those of surface active molecules. It is shown that the nanoparticle-micelle complexes are formed at the interface of an aqueous drop submerged in an oleic phase promoting spontaneous emulsifications and double emulsion formation. Thus, the interface is covered with less than one-micron droplets of emulsified phase. The microemulsion detachment from the interface and their non-uniform distributions trigger fluid circulations inside the aqueous drop. The induced interfacial flow alters the drop shape, contribute in fluid mixing and consequently enhance the emulsification process. We quantify the emulsification rate and the fluid motions at the interfaces and provide plausible physical mechanisms describing the fluid dynamics of the process. We ultimately demonstrate the possibility of generating interfaces with specific microstructures and interfacial properties only by tuning the particle-surfactant concentrations.

3:49PM M04.00030 Particle monolayer growth in evaporating salty colloidal droplets 1, MYRTHE BRUNING, LAURA LOEFFEN, ALVARO MARIN. University of Twente — When pinned colloidal sessile droplets evaporate, the well-known coffee stain effect will occur: particles accumulate along the contact line and form a ring. However, when small amounts of salt are added to the droplet, interesting phenomena occur that alter the particle agglomeration process drastically. As a consequence of the inhomogeneous evaporation along the droplet interface, salt accumulates at the contact line. Since salt increases the surface tension, an interfacial Marangoni flow is generated. This surface flow is directed from the apex of the droplet towards the contact line. It overcomes the bulk capillary flow, resulting in a reversed flow direction as compared to the classical coffee-stain case. Interestingly, despite the flow reversal, particles still accumulate at the contact line. However, in this case particles arrive at the contact line along the liquid-gas interface of the droplet and form a monolayer there. The structure of this particle layer is studied in detail using laser scanning confocal microscopy, which allows us to access the full 3D-position of all particles at the interface and gain insight into their distribution. In this work we will study the dramatic effect that the salt concentration has on the particle layer structure.

1 ERC-StG 678573 Nanopacks

3:50PM M04.00031 Effect of a second component in organic droplet evaporation: initially present versus absorbed during the process, SAHAR ANDALIB, ALI ALSHEHRI, PIROUZ KAVEHPOUR, University of California-Los Angeles — Evaporation of organic liquid has numerous applications ranging from bio-diagnostic to coating technology. Multi-component droplets are often encountered in industrial applications. Even, evaporation of a pure liquid droplet into an environment containing a second substance can result in a multi-component droplet. Despite their omnipresent nature, mechanisms underlying multi-component droplets are not yet fully understood. Present work studies the similarities and differences of the effect of a second component during evaporation of a droplet of an organic solvent. In the first case the second component was present in the droplet from the beginning of evaporation process, while in the second case the second component gets absorbed or adsorbed during the process of evaporation. Analysis of experimental data provides valuable insight into wetting and spreading phenomena of multi-component systems.

3:51PM M04.00032 Using Electrospray to Probe the Interfacial Flow of Evaporating Fluid Masses 1, PAUL CHIAROT, AREF GAFOURI, TIMOTHY SINGER, XIN YONG, SUNY at Binghamton — The ability to measure flow along the surface of an evaporating fluid mass remains a challenge. Tracers are typically dispersed throughout the bulk of the fluid, which means the interfacial transport cannot be easily isolated using particle imaging techniques. In this research, nanoparticles, acting as Lagrangian tracers, are delivered to the interface of sessile drops and rivulets using electrospray atomization. The microdroplets produced by electrospray evaporate in-flight, which leaves behind dry particles that adsorb at the target interface and do not desorb into the bulk. Using this technique, we captured the flow induced on the surface of the fluid masses during evaporation. The flow structure changed dramatically when surfactant was present or if the bulk solvent was a binary (water / alcohol) solution. For these cases, solutal Marangoni flow was induced along the fluid interface, producing flow patterns that were significantly altered from the pure (aqueous) solvent case.

1 This research was supported by the National Science Foundation (Award 1538909).

3:52PM M04.00033 ABSTRACT WITHDRAWN
1. Financial support from CSIR and DST Swarnajayanti is acknowledged.
2. Interdisciplinary Center for Energy Research (ICER), Indian Institute of Science
3. Associate Professor, Department of Mechanical Engineering, Indian Institute of Science

3:55PM M04.00036 Effect of vapor pressure on the performance of a Leidenfrost engine. PRASHANT AGRAWAL, GARY WELLS, RODRIGO LEDESMA-AGUILAR, GLEN MCHALE, Northumbria University, ANTHONY BUCHOUX, KHELLIL SEFIANE, ADAM STOKES, ANTHONY WALTON, JONATHAN TERRY. The University of Edinburgh — Friction is a hindrance to effective mechanical energy conversion, especially at microscales where significant wear occurs due to a high surface-area to volume ratio. Recently, we have developed the concept of a virtually frictionless heat engine, employing levitating liquids (or solids) on turbine-inspired substrates to convert thermal energy to mechanical motion. The levitation is achieved via the Leidenfrost effect, where a liquid drop levitates on its own vapor when it contacts a substrate heated to temperatures significantly above the liquid’s boiling point. The vapor layer virtually eliminates friction and allows evaporating drops to self-propel on asymmetrically textured substrates. In this work, we operate this Leidenfrost heat engine continuously and control its output power by mechanically altering the vapor layer thickness. This is done by replenishing the liquid and supporting the rotor using a bearing assembly. We observe an increase in the power output despite the added bearing friction. The design principles described here can be extrapolated to develop mm and sub-mm scale engines for applications in extreme environments with naturally occurring low pressures and high temperature differences. We acknowledge funding from UK EPSRC (EP/P005896/1 and EP/P005705/1).

3:56PM M04.00037 ABSTRACT WITHDRAWN

3:57PM M04.00038 PIV measurements of a dilute suspension flow over and through various porous media models1, THERESA WILKIE, EILEEN HAFFNER, PARISA MIRBOD, University of Illinois at Chicago — Porous media have become more prevalent in various engineering applications. This study’s aim is to provide insight on how different properties of porous media would affect a dilute suspension flow. Specifically, a suspension fluid containing neutrally buoyant, non-collodial, non-Brownian, rigid, spherical particles with a volume fraction of 3% was examined as it passed over and through various porous media models with porosities of 0.7, 0.8, and 0.9 and thicknesses of 5mm and 3mm. All Reynolds numbers were kept very low to neglect inertial effects. Particle image velocimetry (PIV) technique was used to obtain two-dimensional velocity vectors for planes on top and within the porous media models. To quantify the effect of different porosities and the porous media thickness on a dilute suspension the slip velocity and shear rate analyzed at the suspension-porous interface. We also compared our experimental results with the predicted model, and found a good agreement.

1. National Science Foundation award #1854376

3:58PM M04.00039 Porous walls impact on suspension flows1, SEYEDMEHDI ABTAHI, University of Illinois at Chicago, MARCO ROSTI, KTH Royal Institute of Technology, PARISA MIRBOD, University of Illinois at Chicago, LUCA BRANDT, KTH Royal Institute of Technology — Suspension flows are encountered in various industrial applications including blood flow, transport of slurries, and pharmaceutical industries. We study suspensions of rigid particles in a plane Couette flow with porous walls. To tackle the problem at hand, we perform three-dimensional Direct Numerical Simulations of a plane Couette flow with a suspension of neutrally buoyant rigid particles simulated with an Immersed Boundary Method, while the flow inside the porous walls is simulated using the volume-averaged Navier-Stokes equations. We show that the porous walls produce a shear-thinning effect in the suspensions and this behavior originates from the interactions between the rigid particles, the porous walls and the carrier fluid: non-zero velocity fluctuations at the interface between the porous layers and the clear fluid regions are triggered by the presence of the particles, which in return migrates towards the bulk of the channel. We found that the effect grows with the particle volume fractions and with the permeability of the porous material. At the end for comparison, we also present how the interaction between porous walls and a particle suspension affects a fully developed plane turbulent channel flow.

1. This work has been supported partially by National Science Foundation award 1854376.

3:59PM M04.00040 ABSTRACT WITHDRAWN

4:00PM M04.00041 Versatile particle delivery into dead-end pores enabled by reactive solutes1. XIAOUY TANG, NAN SHI, ANIRUDHA BANERJEE, TODD SQUIRES, UC Santa Barbara — Particle delivery/extraction in dead-end pores is critical in many applications such as drug delivery and oil recovery. However, convective flow is suppressed due to geometrical confinement, while particle diffusion is prohibitively slow. Diffusiophoresis, in which solute concentration gradient drives particle migration, provides a promising strategy. The solute gradient is commonly imposed by introducing a solution of higher or lower concentration and relying on solute diffusion. However, single solute offers very little control over the particle delivery speed. Here, we present a new and versatile strategy to create solute gradient for diffusiophoretic particle migration: chemical reaction. Different model systems are demonstrated including acid-base and precipitation reactions. We will demonstrate that the speed and concentration of the particles can be controlled by varying reactant concentration ratio and reactant diffusivity ratio. Different particle delivery pattern can be achieved: non-focusing or focusing, where particles are delivered in a concentrated band. A theoretical model for the particle delivery dynamics agree well with the experiment. Diffusiophoresis under reactive solute gradient opens up new possibilities to manipulate particle migration in many configurations and applications.

1. Saudi Aramco
4:01PM M04.00042 Transport phenomena in porous structures in interaction with the atmospheric boundary layer , SAURABH SAXENA, NEDA YAGHOOBIAN, Florida State University — This study lays emphases on the underlying physics of mass and heat transfer within porous structures found in the atmospheric boundary layer. The transport process in the porous media happens due to the variation in the structure surface temperature (induced by the variable solar fluxes) and instabilities in the atmospheric boundary layer turbulent flow. The study is inspired by the longstanding problem of the aeration function of soil-based animal-built structures in nature. Direct numerical simulation (DNS) is used to simulate the flow past and through porous media. The Navier–Stokes equations are modified using the Darcy-Brinkman-Forchheimer model to represent the porosity effect. To examine the effect of the natural and forced convection on the transport phenomena within the porous body, the system is further exposed to the diurnal surface temperature variation that is computationally simulated using detailed surface energy balance analyses.

4:02PM M04.00043 ABSTRACT WITHDRAWN —

4:03PM M04.00044 Stress and velocity fluctuations in photoelastic granular avalanches1 , NATHALIE VRIEND, AMALIA THOMAS, University of Cambridge — We study granular avalanches using a custom-built narrow chute where we release 2D photoelastic disks down an incline. Using high-speed imagery, we are able to obtain position and velocity data from particle tracking, and the full stress tensor, including normal and shear stress components, from the photoelastic response of interacting particles. Even though the avalanche is steady-state in time and space, minute fluctuations in velocity and forces away from the mean directly influence the rheology and fluidity. In this study, we analyze the correlation between velocity fluctuations and stress fluctuations in both the quasi-steady layer (close to the rough base) and the flowing layer (near the free surface). We correlate the fluctuations with direct measurements of the measured non-local properties within this photoelastic avalanche.

4:04PM M04.00045 Shear banding and shear jamming in homogeneously sheared granular material1 , YIQIU ZHAO, Department of Physics, Duke University, Durham, NC, 27708, USA, JONATHAN BARÉS, Laboratoire de Mécanique et Génie Civil, Université de Montpellier, CNRS, Montpellier, 34090, France, HU ZHENG, Department of Geotechnical Engineering, College of Civil Engineering, Tongji University, Shanghai, 200092, China, JOSHUA E. S. SOCOLAR, Department of Physics, Duke University, Durham, NC, 27708, USA — We experimentally study the generation and evolution of a shear band during quasistatic shearing of a 2D granular material using a novel split-bottom Couette apparatus in which a layer of photo-elastic disks rests on a base consisting of 21 independently controllable concentric rings. The rings rotate at different rates to generate a uniform basal shear profile. Previous experiments using this setup [arXiv:1904.10051] showed that a steady localized shear band is generated at sufficiently large strains when the packing fraction is higher than a critical value \( \phi_c \approx 0.78 \), which lies between the minimum shear jamming density and the isotropic jamming density. In the present work, we focus on the evolution and structure of the shear band. We find that for the width of the shear band is independent of the global packing fraction above \( \phi_c \), and we analyze the spatial variations in the local packing fraction, the force network structure, and the particle flow field throughout the shearing process.

4:05PM M04.00046 A Simple Photo-resistor sensor for detection and characterization of droplets , MOHAMMAD SHAHAB, ANSHUL VERMA, RAGHUNATHAN PENGASWAMY, Indian Institute of Technology Madras, SYSTEMS ENGINEERING OF THE NATURAL AND THE ARTIFICIAL AT IIT MADRAS TEAM — This work describes a method which utilizes the difference in optical properties (i.e. transmittance) of continuous and dispersed phase to detect droplets in a milli-channel made of PDMS using a sensor i.e. Light Dependent Resistor (LDR) and a light source i.e. Light Emitting Diode (LED). Due to low cost and footprint, LDR is a viable option. LDR also covers the entire spectrum in visible range and gives satisfactory response as opposed to planar diffused photo-diodes which respond only to a limited wavelength-band. Any change in the transmittance of the fluid flowing in the channel changes the intensity of light falling from the LED on LDR resulting in a resistance change in LDR. Droplet detection corresponding to the peaks in the sensor signal due to change in the resistance of LDR is established by comparing the sensor data with the video-processed data. After sensor validation, peak features such as peak height, width and saturation voltage are studied to measure droplet size, velocity and spacing. As an application of this multiphase flow detection, it has been shown that droplets can be classified on the basis of their chemical properties using Beer-Lambert law.

4:06PM M04.00047 Binary Aerosol Composition Measurements using Ultra Small Angle X-ray Scattering1 , DANIEL DUKE, HARRY SCOTT, ANESU KUSANGAYA, DAMON HONNERY, Monash University, BRANDON SFORZO, KATARZYNA MATUSIK, ALAN KASTENGREN, MATTHEW FRITH, JAN ILAVSKY, Argonne National Laboratory, DAVID LEWIS, Chiesi Limited — Nearly all consumer aerosols are binary mixtures of a product in a liquefied propellant. In the pressurised metered dose medical inhaler (PMDI) for example, drugs may be dissolved in ethanol if their solubility in the propellant is poor. The rate of change of liquid composition as the propellant flash-evaporates strongly affects precipitation of the inhaled particles. However, aerosol composition is difficult to measure in an optically dense flash evaporating spray. A novel approach to this problem is demonstrated using Ultra-Small Angle X-ray Scattering (USAXS) on a PMDI solution of 3.36 \( \mu \)g/\( \mu \)L ipratropium bromide, 85% \( \% \)/ R-134a propellant and 15% ethanol. The experiments were conducted at the 9-ID & 7-BM beamlines of the Advanced Photon Source at Argonne National Laboratory. USAXS exploits the high electron density of R-134a relative to ethanol, which leads to a measurable change in X-ray scattering cross-section at the droplet surface. By combining USAXS with X-ray radiography and laser diffraction, the change in cross-section can be measured using Porod’s Law. From this, an ensemble average composition is determined. This new approach enables quantitative validation of binary droplet evaporation models and may lead to improvements in nozzle design.

1Funding: Australian Research Council, DE170100018.
4:07PM M04.00048 Holographic astigmatic particle tracking velocimetry (HAPTV)¹
ZHOU ZHOU, College of Ocean and Atmospheric Sciences, Ocean University of China, SANTOSH KUMAR SANKAR, KEVIN MALLERY, Dept. of Mech. Engineering, UMN, CHENG LI, SAFL, UMN, WENSHENG JIANG, College of Env. Sci. & Engineering, JIARONG HONG, Dept. of Mech. Engineering, UMN — The formation of twin images in digital inline holography (DIH) is the main issue that constrains the placement of the focal plane in the center of the sample volume for DIH-based particle tracking velocimetry (DIH-PTV) with a single camera. As a result, it is challenging to apply DIH-PTV for flow measurements in large-scale laboratory facilities and many field applications despite its low cost and compact setup. Here we introduce holographic astigmatic PTV (HAPTV) by inserting a cylindrical lens in the optical setup of DIH-PTV which generates distorted holograms. Such distortion is subsequently utilized in a customized reconstruction algorithm to distinguish tracers positioned on different sides of the focal plane located in the center of a sample volume. Our HAPTV approach is calibrated under different magnifications, and is implemented to measure a vertical jet flow and the channel flow in a large-scale water tunnel. The results are compared with measurements from conventional particle image velocimetry and show a good agreement. The work has demonstrated that HAPTV can achieve improved spatial resolution compared to the conventional astigmatic PTV, and also enable the implementation of DIH-based PTV to flows in field applications.

¹Chinese Scholarship Council & Ocean University of China

4:08PM M04.00049 Machine learning based holography for flow diagnostics¹, KEVIN MALLERY, SIYAO SHAO, JIARONG HONG, University of Minnesota, Twin Cities — Digital inline holographic particle tracking velocimetry (DIH-PTV) is a promising single camera technique for 3D flow diagnostics due to its low cost, compact setup, versatility, and improved performance as demonstrated in the recent work by Mallery & Hong (Optics Express 27(13), 18069-18084). However, both measurement precision and processing speed remain challenges limiting the broad adoption of DIH-PTV. We present approaches utilizing machine learning to accomplish the particle localization task with both high accuracy and speeds comparable to commercial PIV software. Our approach avoids the need for experimental ground truth measurement by utilizing existing hologram simulation and processing methods to construct a training data set to train multiple convolutional neural networks (CNNs). Our algorithm addresses issues particular to holographic imaging through a novel unification of network elements (including UNET, SWISH activation, and TV loss). We demonstrate that this network can match the results of the best conventional DIH-PTV processing on experimental data and improve speed by a factor of 100. We further demonstrate that this method increases the tracer concentration limits of holography, improving the spatial resolution of measurements.

¹Office of Naval Research No. N000141612755

4:09PM M04.00050 Measuring the dispersion relation of stabilized Faraday waves using Bragg scattering, PAUL W. FONTANA, MASON BROWN, EILEEN FLESHLER, Seattle University — Parametrically excited surface waves, or Faraday waves, are produced on the free surface of a liquid that is vibrated vertically above a critical amplitude. Unlike standing waves, Faraday waves are composed of collections of oscillating solitons, or “oscillons.” These oscillons are generally subject to random motion, but they can be stabilized into a robust square crystalline lattice by dissolving a small amount of a protein, bovine serum albumin (BSA), in the liquid medium. In the experiments presented here, we demonstrate the use of grazing incidence Bragg scattering to measure the lattice spacing of the Faraday wave crystal, and hence the dominant wave number of the Faraday wave. Visible laser light (543.5 nm) is reflected off stabilized Faraday wave crystals at grazing incidence, producing an interference pattern in the far field. Analysis of the interference pattern yields the wave number to high precision and also brackets the wave height. The wave number was measured for driving frequencies from 200 Hz to 900 Hz. The resulting dispersion relation is compared with that of gravity-capillary waves in the same medium. Applications of the Faraday wave crystal as a liquid-based, tunable 2D diffraction grating are being explored.

4:10PM M04.00051 ABSTRACT WITHDRAWN

4:11PM M04.00052 Drag forces of rectangular cylinders with different aspect ratios in a vertical soap film¹, SONG PAN, XINLIANG TIAN, State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University — The soap film provides a convenient way to study the drag force coefficient of a rectangular cylinder in the quasi-2D flow while it has been already investigated with numerical (2D) and experimental (3D) methods. We measured tiny drag forces of different rigid-wire rectangular cylinders by varying their aspect ratios in a vertical flowing soap film. We placed the pre-impregnated rectangular cylinders on a hook inserted into the soap film. The aspect ratios are defined as ratios of the length of cross-stream directions to height in the streamwise. We find that the measured drag forces do not match with published 2D numerical results, indicating the soap film shows 3D features when the aspect ratio is small.

¹Natural Science Foundation of Shanghai(Grant no.19ZR1426300)

4:12PM M04.00053 Gas Jet Blowing on a Falling Soap Film: Three Regimes of Interaction, MAKSIM MEZHERICHER, Princeton University, CEDRIC GERBELOT, Ecole Normale Superieure, ERIC QIU¹, ANTONIO PERAZZO, YIGUANG JU, HOWARD A. STONE, Princeton University — Recently, several studies reported on the generation of soap bubbles by a gas jet blowing perpendicular to a falling soap film. However, an extensive investigation of the regimes of interaction between the air jet and the falling soap film has not been performed yet. In this work, we used hypodermic blunt needles with inner diameters between 110-260 microns as small cylindrical nozzles to create air jets blowing perpendicular to falling soap films with thickness in the range of 2-10 microns, surface tension 25-32 mN/m and viscosity 1-7 mPa*s. While gradually increasing the jet velocity, three regimes of interaction between the blowing jet and the soap film were observed: formation of a dipole in the film for jet velocities smaller than the threshold for bubbling; a bubbling regime over and above a threshold velocity; and a regime of immediate film rupture when a critical jet velocity was achieved. In the bubbling regime, tuning some parameters enabled production of monodisperse bubble aerosols. The onset of the regime of immediate film rupture was found to strongly depend on the nozzle diameter and weakly depend on the film thickness and its physicochemical properties.

¹current affiliation: McKinsey and Company

4:13PM M04.00054 ABSTRACT WITHDRAWN
4:14PM M04.00055 A linear behavior between squeezing pressure and advancing length: when squeezing a viscous droplet into circular narrow confinement at capillary number $= \frac{3}{4}$, CHRISTOPHER LANDRY$^1$, XIAOLIN CHEN$^2$, Washington State University Vancouver, ZHIFENG ZHANG$^3$, Delaware Innovation Campus, American Air Liquide — Particle squeezing through narrow constricted channels occurs in many processes throughout biomedical and chemical engineering fields. Applications range from lab-on-chip devices and pipette aspiration to transport through single pore as well as porous media. Recently, particle behaviors are compared based on particle properties and operation parameters. In the present research, an approximately linear relation between squeezing pressure and advancing length, in which the squeezing of a viscous droplet behaves like a linear solid particle, is found at $\text{Ca}= \frac{3}{4}$. Within the range of theoretical assumption, numerical modeling is conducted to validate the linear relation between the squeezing pressure and advancing length. This finding may have potential implications in cell/particle filtration, aspiration device design, enhanced oil recovery, etc.

$^1$School of Engineering and Computer Science Washington State University Vancouver Vancouver, WA 98686-9600

$^2$School of Engineering and Computer Science Washington State University Vancouver Vancouver, WA 98686-9600

$^3$Process Engineering and Combustion Delaware Innovation Campus, American Air Liquide Newark, DE 19702

Monday, November 25, 2019 4:16PM –
Session NP01 Flash Poster Session I (See M01 for presenter poster order) Exhibit Hall

NP01.00001 Flash Poster Session I (See M01 for presenter poster order) –

Monday, November 25, 2019 4:16PM –
Session NP02 Flash Poster Session II (See M02 for presenter poster order) Exhibit Hall

NP02.00001 Flash Poster Session II (See M02 for presenter poster order) –

Monday, November 25, 2019 4:16PM –
Session NP03 Flash Poster Session III (See M03 for presenter poster order) Exhibit Hall

NP03.00001 Flash Poster Session III (See M03 for presenter poster order) –

Monday, November 25, 2019 4:16PM –
Session NP04 Flash Poster Session IV (See M04 for presenter poster order) Exhibit Hall

NP04.00001 Flash Poster Session IV (See M04 for presenter poster order) –

Monday, November 25, 2019 4:16PM –
Session NP05 Poster Session (4:16pm - 5:16pm) Exhibit Hall

NP05.00001 STUDENT POSTER COMPETITION: THEORETICAL/COMPUTATIONAL –

NP05.00002 Configurations and dynamics of membrane-bound elastic filaments$^1$, WILSON LOUGH, University of Wisconsin-Madison — Changes in the curvature and topology of cell membranes are responsible for numerous biological processes. Many of these changes seem to be driven by interactions with thin filament-like protein structures which form on the surface of membranes. While there are a number of proposed mechanisms, how exactly the filament-membrane interactions produce changes in curvature remains an open question. The feasibility of proposed mechanisms can be be investigated by modeling the filament as a thin elastic rod which is confined to the membrane surface. The interplay between the geometries of the the surface and the filament give rise to complex distributions of force and torque which are believed to play a crucial role in reshaping the membrane. We discuss the mechanics of surface-bound filaments and present a collection of analytical and numerical results.

$^1$NSF: DMS-1661900
NP05.00003 Dynamics of pulsing soft corals\footnote{Funding: NSF PHY grant #1505061}. GABRIELLE HOBSON, LAURA MILLER, University of North Carolina at Chapel Hill, SHILPA KHATRI, University of California, Merced — Soft corals of the family Xenidiidae have a pulsating motion that generates flow in their surrounding fluid. This flow brings new samples of fluid towards the coral, allows sufficiently slow mixing for removal of photosynthetic waste to occur, and then transports the fluid away from the polyp to reduce resampling (Samson et al. 2019). Generating this flow allows the pulsating corals to perform photosynthesis at much higher rates than non-pulsating soft corals (Kremien et al. 2013). Numerical simulations of the pulsations of the coral were conducted using the immersed boundary method. By quantifying flow characteristics such as velocity, vorticity, and Lagrangian coherent structures, we investigated how the flow changes as we varied the frequency-based Reynolds number and the length of the resting period between pulses. Further investigations into the efficiency of fluid transport by the polyps will also be presented. Key words: Immersed boundary method, pulsing soft corals, computational fluid dynamics, biomechanics

NP05.00004 Improving the Foundations for Large Eddy Simulation of Katabatic Flow\footnote{NP05.00007 Energy spectra for turbulent Rayleigh-Bénard convection}. KELSEY EVERARD, University of British Columbia, HOLLY OLDROYD, University of California at Davis, MARCO GIOMETTO, Columbia University, MARC PARLANGE, Monash University, GREG LAWRENCE, University of British Columbia — Large Eddy Simulations (LES) can pose an advantage over Reynolds Averaged Navier-Stokes in that only turbulence at and below the sub-grid scale (SGS) is dictated by a closure model. At the same time, LES can pose an advantage over Direct Numerical Simulations (DNS) in that it can be used to simulate flows at large Reynolds numbers. However, LES results can be sensitive to the choice of SGS turbulence model, especially in the near-surface region of stably stratified flows, making simulation accuracy highly dependent on the near-wall treatment and on grid resolution. For katabatic flow, it is surmised, based on the observed anisotropy and surface-normal flux divergence, that results are sensitive both to the choice of SGS model and to the wall-layer model. However, a mathematically consistent and physically grounded choice does not yet exist in either case, posing a bottleneck for the applicability of LES for katabatic flow simulation. Here, we propose improved models that explicitly account for the physics important to a statically stable fluid flowing under the influence of gravity over a sloped rough boundary. In this way, we contribute to increasing the utility of LES for studies on katabatic flows.

NP05.00005 Three-dimensional bubble mechanics in a corner subject to an acoustic wave. EOIN O’BRIEN, MEHDI MAHMUD, QIANXI WANG, University of Birmingham — The development of cavitation bubbles in a corner subject to an acoustic wave has important applications in ultrasonic cleaning, and cavitation damage. Previous developments have been made on the effects of an acoustic wave on a microbubble near a rigid wall, (K. Manmi 2014), highlighting the effects of the high intensity ultrasound on the bubble’s timespan, jet direction and overall shape of the bubble surface, regarding the initial radius of the bubble and the amplitude of the acoustic wave. Furthermore, numerical simulations have been made to accurately simulate the evolution of a cavitation bubble in a corner created by the intersection of two rigid walls for differing angles. In this presentation, I introduce the mathematical model used for a bubble in a corner subject to an acoustic wave along with the boundary integral method used to numerically model the development of the bubble surface with time. From this, numerical results are presented to highlight the effects of the corner and the acoustic wave for a range of angles, acoustic wave amplitude, and initial bubble radius. In doing so, the jet direction upon collapse can be found, along with the bubble migration towards the corner over time.

NP05.00006 Reliability of general purpose finite-volume solvers for wall-modeled large-eddy simulation of open channel flow at a moderate Reynolds number. WEIYI LI, MARCO GIOMETTO, Columbia University — Wall-modeled large-eddy simulation is increasingly being used in both industry and academia for the characterization of wall-bounded high Reynolds number flows. Numerical simulations are often carried out using general-purpose finite volume solvers, whose solution is known to be particularly sensitive to the nature of the discretization scheme. Such a sensitivity introduces a degree of uncertainty in model results that is yet to be fully quantified. Here we assess the quality and reliability of a general-purpose finite volume solver in wall-modeled large-eddy simulation of a pressure-driven, open-channel flow at frictional Reynolds number 2000. Simulations are performed using an algebraic, equilibrium, wall-layer model. Results are contrasted against corresponding DNS data and predictions from a single-domain pseudo-spectral solver. Model predictions will be discussed with a lens on the impact of the grid aspect-ratio, grid resolution, discretization schemes for the velocity and pressure, time stepping procedure, and velocity-pressure coupling scheme.

NP05.00007 Energy spectra for turbulent Rayleigh-Bénard convection. MICHAEL KWAN, JANET SCHEEL, Occidental College — We investigated the scaling behaviors for numerically simulated, turbulent Rayleigh-Bénard convection by determining the kinetic and thermal energy spectra. The systems have aspect ratio 1, Prandtl numbers 0.7, 0.021 and 0.005, and various Rayleigh numbers ranging from $10^5$ to $10^{10}$. Whereas in previous studies the frequency spectra from a time series were considered, we calculated the energy spectra from spatial fields. In particular, we performed Fourier analysis on two-dimensional cross sections of the temperature and velocity fields. We also computed the conditional velocity and temperature structure functions on the same cross sections to verify our findings. Lastly, we tested whether Kolmogorov’s 1941 (K41) scaling law or the Bolgiano-Obukhov (BO) scaling law applied to both the kinetic and thermal energy spectra for the various systems.

NP05.00008 Using reservoir computing to predict temperature fluctuations in turbulent Rayleigh-Bénard convection\footnote{We thank NSF Awards PHY-1756179 and EAR-1909055.}. SARAH CHANG, Swarthmore College, DANIEL LATHROP, University of Maryland, College Park — Turbulent flows are ubiquitous in the fundamental processes of nature—from the Earth’s mantle to its atmosphere. Among their many complexities, turbulent flows are chaotic, which makes them computationally expensive to simulate and difficult to study analytically. Thus, prediction of turbulent systems is an ongoing challenge. Machine learning methods, specifically reservoir computing, have recently shown promise in predicting turbulent flows. Reservoir computing is a recurrent neural network model that uses a reservoir of randomly connected nodes to process the input data and predict the output for the next time step. As a proof-of-concept, here we test the effectiveness of reservoir computing on the prediction of temperature fluctuations in turbulent Rayleigh-Bénard convection (RBC) because it has a relatively simple experimental setup and relevant applications, like weather prediction. We built a water-filled cylindrical convection apparatus, heated from below and cooled from above, and achieved Rayleigh numbers up to $4.5 \times 10^3$. We train the reservoir computer on our experiments temperature time-series data, collected with thermistors inserted at multiple heights in the flow, aiming to predict temperature fluctuations in turbulent RBC flow.
**NP05.00009 The Hydrodynamic Analysis of Fluid Flow over a Hydrofoil beneath a Free Surface in Subcritical Froude Number.** ZEDA YIN, Graduate student at Marshall University, MEHDI ESMAELPILAR, Assistant Professor at Marshall University — In this work, a numerical simulation of two-dimensional unsteady incompressible viscous flow generated by a shallowly submerged hydrofoil under the free surface is presented. The computation is based on finite volume discretization incorporating the interface capturing volume of fluid method to solve the fluid equation in the motion. The SST k-ω turbulence model is used to capture the turbulent flow for the wave around the hydrofoil. Submerged depths and different angles of attack will be presented in the numerical results. A comparison of the present numerical results with available experiments is performed with the focus on the free surface and flow field for varying AoA and submerged depths.

**NP05.00010 Traveling-Wave Solutions of Two BiDirectional Whitham Equations.** SALVATORE CALATOLA-YOUNG, JOHN CARTER, Seattle University — Following the work of Carter & Rozman (2019), we study traveling wave solutions of two versions of the bidirectional Whitham equation. These equations are extensions of Boussinesq-type models that enable the phase velocities to match those of the Euler equations. We study the systems introduced by Hur & Pandey (HP) (2018) and Aceves-Sanchez et al (2013). We compute families of periodic traveling wave solutions to the HP equation. Our goal is to compare the traveling-wave solutions of both equations to gain further understanding of the differences between them.

**NP05.00011 A Control strategy for Radial Miscible Viscous Fingering: Non Linear Simulations.** VANDITA SHARMA, Department of Mathematics, Indian Institute of Technology Ropar, Punjab India, CHING-YAO CHEN, National Chiao Tung University, Taiwan, MANORANJAN MISHRA, Department of Mathematics, Indian Institute of Technology Ropar, Punjab India — The displacement of a more viscous fluid by a less viscous one in a porous medium results in a hydrodynamic instability called viscous fingering (VF). Chui et al., Phys. Rev. E 92, 041003(R) (2015), experimentally reported the existence of a competition between advective and diffusive forces during the later stages of miscible radial VF. We numerically capture the competition between the aforementioned opposing forces from start to end of the instability. Many attempts have been reported in literature to control VF by utilizing time-dependent strategies, modifying the geometry and altering the fluid properties. However, we utilize the competition to control the instability. Stable displacement is observed despite a less viscous fluid displacing a more viscous one. The M-Pe parameter space is divided into stable and unstable zones by critical parameters following the relation $M_e = Pe_e^{1/2}$. The results are validated by diligently designed experiments.

**NP05.00012 Investigating Nontrivial Time-Periodic Solutions to the Whitham Equation.** CHERIOTHEPHER ROSS, JOHN CARTER, Seattle University — We are interested in waves on shallow water and are studying the existence and evolution of nontrivial time-periodic solutions to the Whitham equation. These solutions can be described as a small-amplitude traveling-wave interacting with a carrier wave. Ambrose and Wilkening (2009) introduced a numerical method for computing such solutions to the existence and evolution of nontrivial time-periodic solutions to the Whitham equation. We study the systems introduced by Hur & Pandey (HP) (2018) and Aceves-Sanchez et al (2013). We compute families of periodic traveling wave solutions to the HP equation. Our goal is to compare the traveling-wave solutions of both equations to gain further understanding of the differences between them.

**NP05.00013 Large Eddy Simulation of Turbulent Coherent Structures in a Storm-forced Surface Ocean Mixed Layer.** CLIFFORD WATKINS, National Center for Atmospheric Research — Submesoscale roll vortices in the ocean mixed layer (OML) are important to understanding and constraining the flux of momentum from the atmosphere into the ocean. In this study, we use large eddy simulations (LES) to investigate the impact of the superposition of Ekman inflection-point (EIP) instabilities on the entrainment and deepening of a shallow coastal ocean pycnocline under the wind forcing of Hurricane Irene (2011). We used reanalysis of the hurricane winds to force LES domains 100m to 4km in the horizontal dimension to observe the development and influence of EIP rolls. The EIP instability has a domain-scale independent wavelength on the order of 500m to a kilometer and transfers momentum to the shear instabilities at the pycnocline, leading to a more rapid deepening and cooling of the OML during the hurricane. By understanding the physical processes driving the changes in the OML both before and during the event, we will be able to predict the development of tropical cyclone intensity over stratified coastal oceans.

**NP05.00014 Three-dimensional visualization of stratified turbulent wakes at varying Reynolds number.** BASEM HALAWA, SHADEE MERHI, CYNTHIA TANG, QI ZHOU, University of Calgary — We present a series of three-dimensional visualizations of numerically simulated turbulent wakes in a stably stratified fluid, with a focus on the effects of the wake Reynolds number, $Re_e$, on the wake flow. The visualization of stratified wakes is complicated by the coexistence of regions of distinct dynamics in the flow, including the large-scale ‘pancake vortices’, small-scale shear instabilities within layers of concentrated vertical shear, and internal waves emitted by the wake to the ambient fluid. We apply various techniques to separate the dynamically distinct regions within a given wake and visualize them respectively. The volume fractions occupied by each of these regions are also quantified. Through the visualizations, we observe several significant effects on the wake’s evolution associated with increasing wake Reynolds number, which will be presented in the poster.

**NP05.00015 Providing biomimetic passive pumping on an implantable biosensor.** ALEC DRYDEN, MATTHEW BALLARD, Saint Martin’s University — Microfluidic technologies have opened doors to rapid, inexpensive and compact medical diagnostic testing, with one important example being the glucose meter. Technology continues to evolve to continuous compact medical diagnostic testing, with one important example being the glucose meter. Technology continues to evolve to continuous monitoring through mounting microfluidic devices onto the patient. A major hurdle in the development of on-body sensors is that they typically require flow of bodily fluids such as interstitial fluid or blood through the device. As a result, on-body sensors are typically bulky, requiring a pump and power source. We propose a solution for passive pumping on an implantable biosensor by mimicking nature’s elegant solution to the problem – lymphatic vessels. The lymphatic system pumps fluid using unidirectional flexible valves which open and close with fluctuations in fluid pressure, driven by contraction and expansion of surrounding muscle and of the lymphatic vessels. The ability of easily manufacturable bioinspired polydimethylsiloxane (PDMS) bioinspired valves, anchored to the microchannel floor, to mimic lymphatic valves, opening and closing to allow forward flow and prevent backflow, respectively. The design’s size scale, biocompatible materials, and reliance on biologically driven pressure gradients allow implantability and defeat the need for active pumping from an external source.

1This material is based upon work supported by the National Science Foundation under grant DMS-1716120.

1The support from NSERC, Compute Canada and MEOPAR is gratefully acknowledged.
NP05.00016 Multiplicity in Stable Orbits for Prolate Capsules in Simple Shear Flow 1, XIAO ZHANG, MICHAEL D. GRAHAM, University of Wisconsin-Madison — Artificial capsules have been applied in numerous fields such as bioengineering, pharmaceutics, and food industry. This work investigates the motion of a deformable prolate capsule in unbounded simple shear flow at zero Reynolds number using direct simulations. The deformability, bending stiffness, initial orientation, aspect ratio of the capsule and the viscosity ratio between the inner and outer fluids are varied over a wide parameter space. At a low viscosity ratio, a capsule with large bending stiffness always tends towards an in-plane stable orbit, either tumbling or swinging, depending on the deformability, i.e., capillary number (Ca). At a high viscosity ratio, however, a tumbling-to-rolling transition is observed for a capsule with large bending stiffness at increasing Ca. In the transition regime, the capsule is found to adopt multiple stable orbital modes including tumbling, precessing and rolling, depending on the initial orientation. This multiplicity regime becomes broader as the aspect ratio of the capsule increases, while showing an opposite dependency on the viscosity ratio. The multiplicity in stable orbits leads to a multiplicity in the rheological behavior for a dilute suspension of such capsules.

1This work is supported by NIH grant 1R21MD011590-01A1.

NP05.00017 ABSTRACT WITHDRAWN —

NP05.00018 Water Desalination with Two-dimensional Metal Organic Framework Membranes, ZHONGLIN CAO, VINCENT LIU, AMIR FARIMANI, Carnegie Mellon University — Providing fresh and drinkable water is a grand challenge the world is facing today. Development in nano-materials can create possibilities of using energy-efficient nanoporous materials for water desalination. In this work, we demonstrated that ultrathin Metal Organic Framework (MOF) is capable of efficiently rejecting ions while giving access to high water flux. Through molecular dynamic simulation, we discovered perfect ion rejection rate by two-dimensional multi-layer MOF. The naturally porous structure of 2D MOF enables significantly 3 to 6 orders of magnitude higher water permeation compared to that of traditional membranes. Few layers MOF membranes show one order of magnitude higher water flux compared to single layer nanoporous graphene or molybdenum disulfide without the requirement of drilling pores. The excellent performance of 2D MOF membranes is supported by water permeation calculations, water density/velocity profiles at the pore and the water interfacial diffusion near the pore. Water desalination performance of MOF offers a potential solution for energy-efficient water desalination.

NP05.00019 Effects of Implant Separator Structure on Drug Delivery to the Posterior Eye,1, SEYEDALIREZA ABOOTORABI, Indiana University-Purdue University Indianapolis, ABHI TRIPATHI, LILIAN DAVILA, University of California, Merced, HUIDAN YU, Indiana University-Purdue University, Indianapolis — The prevalence of visual impairment and blindness attracts attentions to develop new types of implant for drug delivery that can reduce drug administration doses and regulate drug release rates. We use Comsol Multiphysics to simulate drug transport from a hydrogel implant behind the sclera layer to the posterior eye. The focus is on the time evolutions of the drug concentration in the sclera, choroid, and retina layers respectively. The computational domain and dimensions are from a prior study (Kavousanakis et al. 2014). The governing equations are coupled between incompressible Darcy’s law for flow and diffusion-advection for concentration. Drug delivery from the hydrogel implant directly agrees with open data. A new polymeric implant, containing several compartments defined by porous separators, is then introduced between the hydrogel and sclera to regulate the drug delivery rate. It is found that the peak concentration appears at approximately the same time but with much larger magnitude with no separator vs. with zero blockage separator. When the blockage widths in the separator is increased, the peak concentration appears in later time and smaller value. Such a new implant seems to be a plausible alternative to the drug delivery implants available to date in the market.

1This work is supported by IU-MSI seed fund

NP05.00020 Suspension flows in a pipe covered with permeable surfaces1, MARYAM BAGHERI, CHANGWOO KANG, PARISA MIRBOD, University of Illinois at Chicago — The flow of particulate suspensions driven by a constant pressure gradient is examined in a pipe where its walls are replaced with a permeable surface. We explore non-colloidal suspensions of rigid and spherical particles in a Newtonian fluid over a wide range of bulk particle volume fraction (0.1 ≤ φb ≤ 0.5) and the permeability of the porous medium. Direct numerical simulations (DNS) are performed to solve conservation equations of the flow coupled with the constitutive equation of suspensions (Diffusive Flux Model) and Darcy’s law in a porous medium. We aim to elucidate the effect of the permeable surface on the suspension flows. The velocity and concentration profiles are presented for various control parameters and show that the velocity of suspensions enhances by the slip effect at the suspension-porous interface. We evaluate the rate of suspension flows and slip velocity at the suspension-porous interface induced by the permeable surface. It reveals that the rate of suspension flow decreases as the bulk volume fraction φb increases and it builds up as the permeability of the porous media increases. We also show that there are two different regimes characterizing the slip velocity normalized by both shear rate and penetration depth, at the suspension-porous interface, namely the strong permeability regime and the weak permeability regime.

1This work has been supported partially by National Science Foundation award #1854376.

NP05.00021 Developing Notebook-based Flow Visualization and Analysis Modules for Computational Fluid Dynamics1, GERMAN SALTAR, University of Puerto Rico - Mayaguez, ADITYA AIYER, CHARLES MENEVEAU, Johns Hopkins University — Visualization techniques are essential in identifying complex features and patterns in fluid flows. Three dimensional visualization has long been confined to high-end software. Recently, packages for Python have been developed to overcome this limitation. We seek to implement them in the realm of fluid mechanics using Jupyter notebooks. To this end, we make use of Python’s J3D-Jupyter package to generate 3D volume rendering from Large Eddy Simulation (LES) datasets. Using volume rendering, we were able to visualize velocity and concentration fields of oil droplets of varying sizes injected at the centerline of a turbulent jet influenced by a uniform crossflow. Alternately, uniformly random spheres modulated by the LES computed concentration field were generated to represent the Eulerian concentration field. Interactive features allowed the 3D structure of the jet to be probed and the turbulent structure’s role in the oil’s spatial distribution to be inferred. Additionally, we were able to implement the modules to visualize Direct Numerical Simulation (DNS) data obtained from the Johns Hopkins Turbulence Data Base (JHTDB). The capability of exporting the interactive plots as html files, which can be embedded into a website or online research article, facilitates distribution.

1National Science Foundation (grant # OCE-1633124), Gulf of Mexico Research Initiative, JHU’s Whiting School of Engineering
NP05.00022 Inferring an effective eddy viscosity from High Fidelity Data. NIKHIL OBEROI, WALTER ARIAS RAMIREZ, JOHAN LARSSON, University of Maryland, College Park — Different ways to compute an inferred eddy viscosity from resolved turbulence data (from LES, DNS, experiments) based on optimization methods are investigated and assessed based on how well they reproduce the mean fields. The method is tested on wall bounded flows including a channel and a boundary layer. We further investigate the sensitivity of these methods to averaging error in the given Reynolds stress fields.

1This work has been supported by AFOSR grant FA9550-19-1-0210

NP05.00023 Clustering of Vortex Wakes for Heaving-Pitching Foils Using Machine Learning, MUKUL DAVE, ALEJANDRO GONZALEZ CALVET, JENNIFER FRANCK, University of Wisconsin-Madison — A heaving-pitching foil produces different vortex patterns that are associated with propulsive regimes of high efficiency or high thrust, however there are many intermediate modes not readily classified due to a chaotic wake structure. Hence, machine learning techniques can be applied to help cluster wake patterns and associated performance. To evaluate the achievement of machine learning, a database of Reynolds-averaged Navier-Stokes simulations at Reynolds number of 10^6 and high heave amplitudes is utilized. The data includes sweeps in flapping frequency and pitch amplitude to produce a wide range of kinematics and propulsive modes. A convolutional autoencoder with lasso regularization was used to extract a latent space of important features from vorticity images of the wakes. Applying different clustering algorithms to the latent space groups the data into distinct flow regimes, such as a low vorticity regime with minimal flow separation and a high vorticity regime with a reverse von Kármán wake. The results are validated against a pre-labeled subset of samples to explore ways of improving the clustering. To correlate wake clustering with performance, support vector regression was used to predict the values of efficiency, power and thrust based on the kinematics.

NP05.00024 Simulation of nonlinear ocean waves using volume-of-fluid method, ZHOU ZHANG, KEVIN MAKI, YULIN PAN, Department of Naval Architecture and Marine Engineering, University of Michigan — The volume-of-fluid (VOF) method is widely used in the numerical simulation of multi-phase flows. In this work, we investigate the capability of VOF method to model nonlinear ocean waves, using several algebraic and geometric interface-capturing approaches. By considering a low viscosity and an initially irrotational flow field, the VOF solution can be validated against a fully nonlinear potential flow solution. We perform this study for both regular and irregular nonlinear waves, which benchmarks the capability of VOF to simulate the evolution of ocean waves. Furthermore, we develop a nonlinear wave inlet boundary based on the relaxation zone technique, enabling the simulation of prescribed incoming nonlinear waves into the computational domain. These developed capabilities are expected to be beneficial for different scenarios in ocean engineering.

NP05.00025 Numerical modeling of prostatic artery embolization: patient-specific blood flow and emboli transport, CHADRICK JENNINGS, MOSTAFA MAHMOUDI, Northern Arizona University, ANDREW HALL, Saint Louis University, AMIRHOSSEIN ARZANI, Northern Arizona University — Benign prostatic hyperplasia (BPH) is the most common non-cancerous tumor found in men. Symptoms caused by BPH can be treated by a recently proposed minimally invasive procedure known as prostatic artery embolization (PAE), where particles are injected through a catheter to limit blood supply to the enlarged prostate. The goal of this study is to characterize the complex blood flow patterns in common iliac and prostatic arteries and to model PAE with particle tracking in a patient-specific model. A computer model of common iliac arteries and the downstream vasculature was created from CT-angiography images using SimVascular. Image-based computational fluid dynamics (CFD) simulation using Oasis (a minimally dissipative open-source solver) was performed to obtain velocity data. Finally, the Maxey Riley equation was solved to model the PAE procedure and guide embolus injection. Our patient-specific computer model can provide valuable information for the PAE procedure.

NP05.00026 Wall shear stress manifolds and surface temperature patterns in heat transfer enhancement applications, CONNOR MORENO, AMIRHOSSEIN ARZANI, Northern Arizona University — Heat transfer enhancement has applications in many of the devices used by the public, directly or indirectly, on a daily basis. However, high-resolution heat transfer simulations are computationally costly, representing significant downtime in engineering industry and research. Meanwhile, wall shear stress (WSS) topology has been recently demonstrated as an effective estimator of surface concentration patterns in convective mass transport settings. Particularly, stable and unstable manifolds in WSS have been shown to control near-wall transport patterns in cardiovascular flows. Estimating surface concentration/temperature via WSS is desirable because of the significant reduction in computational time and the physical explanation of the results. In this study, it is investigated whether WSS topology proves an effective estimator of surface temperature patterns in heat transfer settings. The configuration used is the classical impinging jet, a widely used heat transfer enhancement application. WSS is obtained by computational fluid dynamics simulations and surface temperature is obtained by solving the advection-diffusion equation, both using the finite element method. Our results demonstrate a close connection between WSS manifolds and surface temperature patterns.

1Funding from the NAU NASA Space Grant Fellowship is acknowledged.

NP05.00027 2D CFD Analysis of NASA X-57 Maxwell High Lift Propeller Airfoil, SUSAN SANTIAGO, City College of New York, JINWEI SHEN, KYLE NELSON, The University of Alabama — The X-57 Maxwell is an experimental aircraft developed by NASA. This aircraft is designed to be quieter, lighter, and more efficient compared to the aircraft it is based on, the Tecnam P2006T. The aircraft has two large, outer propellers and 12 small, high lift, collapsible inner propellers. The high lift propellers are only active during low-speed flight which includes take-off and landing. During high-speed flights, the high lift propellers collapse into the wing of the aircraft and are no longer active. The goal of this aircraft is to demonstrate the benefits of distributed electric propulsion (DEP). Our research aims to calculate aerodynamic coefficients including lift, drag, and pitching moment on the airfoil of the high lift propellers. The coefficient tables generated will be used to obtain the aerodynamics for multi-body dynamics simulation testing of whirl flutter stability on the aircraft. To perform the computational fluid dynamics (CFD) calculations, we first conducted a validation study on a NACA 0012 airfoil. Using the same methodology, a CFD analysis was conducted on the high lift airfoil. The analysis on both the NACA 0012 and high lift airfoil were done using both Fluent and Genesis in order to validate the results obtained.

1NSF Grant: 1659710
NP05.00028 Molecular Dynamics Analysis on the Dynamic Contact Angle Based on the Extraction of the Stress Distribution in Steady-State Non-Equilibrium Systems of a Single Lennard-Jones Fluid between Solid Walls under Shear, HIROKI KUSUDO, TAKESHI OMORI, YASUTAKA YAMAGUCHI, Department of Mechanical Engineering, Osaka University — The Method of Plane (MoP) is a technique to calculate the stress distributions for systems in equilibrium molecular dynamics (EMD) simulations. In this study, we propose a method to extend the MoP to steady-state nonequilibrium molecular dynamics (NEMD) systems based on the velocity distribution function. Moreover, we examined the momentum balance exerted on a control volume set around the contact line for a single fluid between parallel walls under steady shear. By extending Bakkers equation, which connects the stress distribution and solid-related interfaces, we showed that the force balance among the dynamic interfacial tensions around the contact line can be rewritten by a model equation equivalent to Young's equation for equilibrium systems. We applied the model equation for steady state NEMD systems of single Lennard-Jones fluid by calculating the stress distribution, and showed that for the present system, the dynamic solid-liquid interfacial tension was constant sufficiently away from the contact line, and the constant value was almost equal to the static value irrespective of the shear rate tested. This result indicated that the dynamic apparent contact angle was not significantly different from the equilibrium apparent contact angle for the present system.

NP05.00029 Heat Transfer Analysis and Heat Sink Design for MELD Additive Manufacturing, JACOB STAFFORD, University of Evansville, DAVID MACPHEE, PAUL ALLISON, University of Alabama — MELD manufacturing is a solid-state additive manufacturing process that provides an alternate path to fusion-based additive manufacturing. Understanding the thermal cycle and hardness of the weld zone for components made using the MELD manufacturing process are essential in the quality of the manufactured components. To facilitate better understanding of the process, a simulation of the conjugate heat transfer problem was modeled in ANSYS Fluent. The model was then used to design a heat sink to better disperse thermal energy during material deposition. As a baseline for the heat sink design, a cold plate used in HVAC applications was modeled and tested experimentally. The computational solver was refined using grid size and time-step independence tests, and validated with experimental data. This computational solver was then used to design a new heat sink. By controlling the flow of the deposited material, better predictions for the thermal cycle and hardness are possible. This allows for improved prediction of the quality of solid-state additive manufacturing components.

NP05.00030 Understanding the dynamics of reservoir computing in chaotic dynamical systems, ADAM SUBEL, ASHESH CHATTOPADHYAY, PEDRAM HASSANZADEH, Rice University — Data-driven prediction of chaotic and multiscale dynamical systems is a daunting task. Recently there has been a lot of interest in predicting chaotic and turbulent flow using deep learning methods which have shown mixed results. However, reservoir computing algorithms such as echo state networks (ESN) have shown promise in capturing this behavior both in short term direct prediction as well as the long-term statistics of chaotic systems better than the state-of-the-art in recurrent neural networks. ESNs are an appealing choice for data driven prediction since unlike deep learning algorithms that train the weights through an expensive backpropagation algorithm, the ESN only trains a layer of output weights with linear regression making it orders of magnitude cheaper to train. The memory of the network comes from updating a state vector in the reservoir, using the previous state and the input to the system. In this work we present a theoretical understanding of reservoir dynamics and how it learns the behavior of chaotic signals and how the signals split up as the “echoes” of the system, an analytical expression and bound for the “dynamical memory” in the reservoir and seek to understand the mechanism which brings about the effectiveness of ESNs in capturing chaotic dynamics.

NP05.00031 Non-negligible Flow Scale for the Vortex Generation in an Isotropic Homogeneous Turbulence, SHO SAEKI, Division of Mechanical Engineering, Graduate School of Engineering, Aichi Institute of Technology — The present study investigates vortical structures in an isotropic homogeneous decaying turbulence in the Direct Numerical Simulation. Subjected nodes of the vortex generation are extracted by monitoring the swirlity that specifies the unidirectionality and intensity of the azimuthal flow. In the transition, middle and small flow scales, e.g., the flows composed of wavenumbers 20-40 and 60-120 in the total wavenumber less than 120, are indispensable for the creation of the key flow. Therefore, although much smaller flow scales have less kinetic energy and may be assumed not to affect greatly to turbulent flows in terms of flow dynamics, these may play non-negligible role for the vortex generation. This feature seems to be inherent in low Reynolds number of this turbulence.

NP05.00032 Analysis of Vortical Structure in Terms of Local Flow Topology in an Isotropic Homogeneous Turbulence, DAIKI AOYAMA, Graduate School of Engineering, Aichi Institute of Technology — The present study investigates vortical structures in an isotropic homogeneous decaying turbulence in the Direct Numerical Simulation. We apply the swirl plane and physical quantities, the swirlity, soriuity and vortical flow symmetry quantity, associated with the invariant local topology (Nakayama, FDR 2014). The swirlity and soriuity specify the unidirectionality and intensity of the azimuthal flow and radial flow, respectively in an arbitrary plane, and the symmetry quantity specifies the direction and the degree of skewness of the vortical flow. The vortical structure is statistically analysed in terms of the distribution of azimuthal velocity and radial velocity which are extracted from the flow in the swirl plane around points where the swirlity has maximum value after respecting the direction of skewness of the vortical flow in that plane because vortical flow generally swirls with respective skewness. The swirling radius is estimated from the distribution of both azimuthal velocity and swirlity. The distributions of azimuthal velocity and radial velocity that are non-axisymmetrically and non-linearly, respectively, indicates that none of vortex models is similar to this vortical structure.

NP05.00033 Relationships between the eigen-vortical-axis line and the vortex stretching in an isotropic homogeneous turbulence, HAYATO HORI, Graduate School of Engineering, Aichi Institute of Technology, KATSUYUKI NAKAYAMA, Department of Mechanical Engineering, Aichi Institute of Technology — The present study investigates the relationships between the eigen-vortical-axis line and the vortex stretching in an isotropic homogeneous decaying turbulence in a low Taylor Reynolds number. This axis line has been proposed as a vortical axis based on the invariant local flow topology and shown that it has stronger characteristics to pierce intense vortical regions than the vorticity line. The vortex stretching can be classified into (1) ineffective stretching that increases vorticity parallel to the swirl plane and (2) effective stretching that increases the orthogonality of the vortical axis and develops a vorticity component associated with swirling. The characteristics of the vortex stretching in a vortical axis are analysed focusing on the effective stretching and an angle between the directions of the vortical stretching and the local axis. Furthermore, it is analysed that the influence of the vortex stretching to the local axis geometry that is defined by the eigenvalues in terms of the gradient of the local axis direction and shows the behavior of the axis line, e.g., twist and convergence. These analyses show that the eigen-vortical-axis line has higher correlation with respect to the effective stretching than the vorticity line.
NP05.00034 ABSTRACT WITHDRAWN

NP05.00035 Qualitative Approaches to Understanding Coherent Structures in Turbulence
IHSAN T. SHAIFI, Department of Nuclear Engineering, Texas AM University, PAUL J. KRISTO, Department of Mechanical Engineering, Texas AM University, ABDULLAH G. WEISS, Department of Nuclear Engineering, Texas AM University, MARK L. KIMBER, Department of Mechanical Engineering, Department of Nuclear Engineering, Texas AM University — A review of several techniques is presented, with emphasis on recent developments in the mathematical treatment of the proper orthogonal decomposition (POD) as applied to experimental particle image velocimetry data. The experiment in question is the classical flow past a cylinder with three distinct shapes: round, square, and hexagonal at a Reynolds number of 16000. The purpose of the study is to analyze the identification of basic structures and estimation of energy content via filtering methods. Reynolds decomposition is the method of separating the flow field into its mean and fluctuating components. Galerkin decomposition uses the bulk velocity to identify small-scale vortices within the flow field. LES decomposition is analogous to a low pass filter that reveals small length scales. POD is an advanced method that takes high dimensional nonlinear coherent structures and converts them into a finite dimensional linearized projection. Vorticity is used to describe the local spinning motion near an eddy and to recover a velocity field from a vorticity field. The accuracy of each method is compared and differences in structures produced by each geometry are discussed. Practical concerns include computational time, data compression, and spatial resolution are addressed.

NP05.00036 Modeling Drop Carrier Particles Through Energy Minimization
BERNANDO HERNANDEZ ADAME, Massachusetts Institute of Technology, RYAN SHIJIE DU, University of California-Los Angeles, LILY LIU, University of Chicago, SIMON NG, University of California, Los Angeles, HANSSELL PEREZ, University of California, Merced, SNEHA SAMBANDAM, KYUNG HA, CLAUDIA FALCON, DINO DI CARLO, ANDREA BERTOZZI, University of California, Los Angeles — Drop Carrier Particles (DCPs) are solid microparticles designed to capture uniform microplet drops of a target solution without using costly microfluidic equipment and techniques. DCPs are useful for automated and high-throughput biological assays and reactions, as well as single cell analysis. However, little work has been done to understand the behavior of these particles in large scale systems, and researchers have had difficulty achieving uniform volume across the particles. Here we present a method for modeling a diverse range of geometric particle shapes using energy minimization techniques. Furthermore, interactions between two particles are modeled as a pairwise process of minimal energy splitting of the target fluid. We examined the effects of particle geometries on the expected number of interactions needed to achieve the desired uniform volume distribution. We then performed macro-scale experiments of two-particle interactions and compared the observed splitting behavior with our simulations. These comparisons indicate that the model accurately predicts particle splitting behavior and multi-particle volume distributions, thus providing engineers with insight regarding optimal particle geometries.

NP05.00037 Effects of Asymmetries on the Evolution of an Indirectly Driven ICF Capsule Outer Shell
CALVIN YOUNG, Los Alamos National Lab, UNIV. OF MICHIGAN COLUMBIA, ERIC LOOMIS, PAUL KEITER, Los Alamos National Lab, P-24, TANA CARDENAS, Los Alamos National Lab, MST-7, CASEY KONG, General Atomics, JACOB MCFARLAND, Univ. of Missouri Columbia — Nuclear fusion offers clean power production in a compact form, and as such is a focus of many avenues of research. Inertial Confinement Fusion (ICF) is particularly promising, though many hurdles remain in the attainment of fusion experimentally. ICF methods involve imploding a spherical capsule composed of hydrogen fuel sheathed in layer(s) of specially selected materials. The implosion is driven indirectly by bathing the capsule in laser driven x-rays. The ionized outer shell is forced inwards by ablative force, compressing the inner layers as a piston, until the fuel reaches an energy state at which robust burn and fusion reactions occur. Fabrication of the shell is difficult, and the form is an imperfect spheroid. Imperfections lead to asymmetrical evolution as the outer shell implodes, introducing instabilities which reduce compressive efficiency. It is necessary to be able to characterize the shape of the outer shell, to predict with simulations the effect of surface features on evolution during implosion. In this presentation I discuss the development of a tool which reads manufacturer profile data of the capsule surface and returns orientation and spherical harmonics. This data was then used to determine the evolution of features during initial implosion stages using an ablative rocket model. Results from these calculations can be used to calculate growth factors for instabilities such as the ablative Rayleigh-Taylor.

NP05.00038 Frequency Downshift in the Ocean
CAMILLE R. ZAUG, JOHN D. CARTER, Seattle University — Frequency downshift (FD) occurs when a measure of a waves frequency (typically its spectral peak or spectral mean) decreases monotonically. Carter et al. (2018) compared the efficacy of generalizations of the nonlinear Schrödinger equation (NLS) at modeling waves with and without FD in wave tank experiments. Narrow-banded swell traveling across the Pacific Ocean can also undergo FD, as evidenced in the classic experiments of Snodgrass et al. (1966). In this work, we compare (i) NLS, (ii) dissipative NLS, (iii) the Dysthe equation, (iv) the viscous Dysthe equation, (v) the dissipative Gramstad-Trulsen equation, and (vi) the Islas-Schober equation to see which model best describes the ocean data reported in Snodgrass et al., regardless of observed FD. We do so by comparing the Fourier amplitudes, spectral peak, spectral mean, and quantities representing mass and momentum between the ocean measurements and numerical simulations.

NP05.00039 Accuracy of Equations Modeling Higher Harmonics in Surface Water Waves
HANNAH POTGIETER, JOHN CARTER, Seattle University — We study the evolution of the higher harmonics in surface water wave experiments. We compare numerical predictions from asymptotic reductions of the Euler equations and its dissipative generalizations with measurements from water wave experiments conducted at Penn State University. Our models include the (i) nonlinear Schrödinger equation (NLS), (ii) dissipative NLS equation, (iii) Dysthe equation, (iv) viscous Dysthe equation, and (v) the dissipative Gramstad-Trulsen equation. We find the predictions from these models are not always consistent with the experimental data.

NP05.00040 Modeling Passive Drag-Based Fish Interactions and their Relation to Formation Behaviors
ABDRAHMAN MANSY, IMRAAN FARUQUE, Oklahoma State University — Recent work coupled high speed imagery-based fish schooling measurements with computational fluid dynamics to provide estimates of the fish’s thrust and drag forces. Reconfigurations indicating reductions in body drag during periods with minimal changes in kinematic inputs suggest that passive drag-based mechanisms could play a role in school reconfiguration and reduce the on-board sensory feedback demands. We use simplified interaction models to study the effects of passive drag-based interaction mechanisms in biological formations. Incompressible hydrodynamic forces were modeled as 1-D incompressible functions of egomotion, position, and velocity. We find conditions for relative position stability within the formation and compare two cases: (a) egomotion and relative position sensitivity only, and (b) egomotion, relative position, and relative velocity. Model (a) shows the agents’ relative positions are dynamically unstable theoretically and in simulation, while (b) is dynamically stable. This finding suggests that mechanisms to reconfigure individuals to lower drag states require relative velocity sensitivity, either through fluid interaction functions or active sensory feedback.
NP05.00041 Analysis of the flow of grains through a screw conveyor

AASHISH GUPTA, PRABHU NOTT, Indian Institute of Science — Screw conveyors are widely employed in industry for the bulk transport of particulate materials. Several studies have attempted to correlate the discharge rate with the angular velocity of the screw and the pitch to diameter ratio via experiments and particle dynamics simulations. However, a detailed mechanical model that would assist in optimal design of screw conveyors, hasn’t been attempted. In this study, we first construct a simple model that assumes the entire granular medium to move as a rigid body sliding along the surfaces of the screw and casing. By enforcing the balances of linear and angular momentum to a suitably chosen continuum element, we show that under certain limiting conditions, the discharge rate for a given angular velocity and screw geometry can be obtained. Further, the discharge can be maximized by setting the pitch to casing radius ratio to a particular value. We then study the detailed flow within the conveyor using the discrete element method and show that a significant fraction of the material exhibits solid body motion, in agreement with the simple model. We assess the effect of relaxing the limiting conditions employed in the model, thereby determining the connection between the friction at the walls and the discharge rate.

NP05.00042 A Yang-Mills Approach for Conformal Invariance in Turbulence

ANTONINO TRAVIA, RAZVAN TEODORESCU, University of South Florida — The relationship between conformal symmetry and stochastic processes has been a rapidly growing field over the last twenty years. Among other applications, this pairing was used to suggest the presence of conformal invariance in two-dimensional turbulence via numerical methods. We present a theoretical approach to realize this correlation for an incompressible fluid of the same dimension using techniques from Yang-Mills theory with a non-Abelian gauge group. In doing so, we also provide a new explanation for behavior near zero-vorticity lines.

NP05.00043 STUDENT POSTER COMPETITION: EXPERIMENTAL –

NP05.00044 Asymmetric forcing of two coupled thermoacoustic oscillators

YU GUAN, LARRY K.B. LI, The Hong Kong University of Science and Technology — In many combustion systems, such as gas turbines and domestic boilers, the presence of self-excited thermoacoustic oscillations can reduce reliability and efficiency. Recent studies have shown that such oscillations can be eliminated by carefully tuning the dissipative and time-delayed coupling between adjacent combustors, exploiting a nonlinear phenomenon known as amplitude death (AD). However, although the coupling conditions required for AD are well known, they may still be beyond the reach of practical systems because of the inherent space and operational limitations that such systems face. To address this issue, we examine whether external forcing can be used to enlarge the AD boundaries of a coupled thermoacoustic system. The system consists of two self-excited thermoacoustic oscillators interacting with each other via tunable acoustic dissipative and time-delayed coupling. By varying the strength of the forcing acting on each oscillator and the phase difference and detuning between the two forcing signals, we map out the forcing conditions required for AD under different initial coupling conditions, paving the way for an alternative method of suppressing harmful thermoacoustic oscillations in combustion systems.

1This work was funded by the Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815).

NP05.00045 Closed-loop control of thermoacoustic oscillations using genetic programming

ANIMESH KUMAR JHA, BO YIN, LARRY K.B. LI, Hong Kong University of Science and Technology — The use of genetic programming (GP) to discover model-free control laws for nonlinear flow systems has gained considerable traction recently, having been applied for the closed-loop control of recirculation zones behind backward-facing steps, flow separation over sharp edges and turbulent mixing layers. This unsupervised data-driven control strategy has been shown to outperform conventional open-loop forcing, by enabling successful individual control laws to spread their genetic traits from one generation to the next. In this experimental study, we use GP to discover model-free control laws for the suppression of self-excited thermoacoustic oscillations, which are detrimental to combustion systems. We evaluate every individual control law in a given generation on a real-time closed-loop control system equipped with a single sensor (a pressure transducer) and a single actuator (a loudspeaker). We rank the effectiveness of the control laws with a cost function and use a tournament process to breed subsequent generations of control laws. We then benchmark the performance of the final generation against that of open-loop forcing, providing improved control laws for the suppression of self-excited thermoacoustic oscillations.

1This work was funded by the Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815).

NP05.00046 Aerodynamic characteristics and flight behavior of the turbo-jav in the javelic throw

HARUKI NAKAYAMA, Graduate School of Kansai University, TOMOYA NAKAJIMA, Osaka Prefecture University, TOMOAKI ITANO, Kansai University, SUGIHARA-SEKI MASAKO, Kansai University, Osaka University — In order to improve the record of the javelin throw in the junior Olympic games, it is important to elucidate the flight characteristics of the “turbo-jav” used in this throwing event. The turbo-jav has many geometrical features different from a spear for the javelin throw, including the presence of tail fins. In the present study, we performed wind tunnel tests in a low speed wind tunnel to measure the drag force, lift force and pitching moment exerted on the turbo-jav for various angles of attack. Using the aerodynamic coefficients obtained, we numerically calculated the flight orbit of the turbo-jav for various angles of attack. Using the aerodynamic coefficients obtained, we numerically calculated the flight orbit of the turbo-jav for various angles of attack. This work was funded by the Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815).

NP05.00047 Using quadrotor IMU data to estimate wind velocity

MEGAN MAZZATENTA, DARIUS CARTER, DANIEL QUINN, University of Virginia — Due to their hovering ability and light frame, Micro Aerial Vehicles (MAVs) have the potential to conduct cheaper wind measurements with greater spatial resolution than weather balloons. However, wind sensors increase MAV payload and therefore increase cost and decrease endurance. To avoid a mounted sensor, wind velocity estimation models have been constructed using on-board Inertial Measurement Unit (IMU) data collected during flight. Existing models are able to relate IMU data to wind velocity, but they rely on calibrations and physical assumptions that limit measurement accuracy. To improve the accuracy of velocity measurements, we used a quadrotor to collect IMU data while surveying the surrounding flow using Particle Image Velocimetry (PIV). We compared IMU and PIV data for a quadrotor in still air, in the wake of another quadrotor, and in a vortex. We then evaluated how well existing models could determine velocity and turbulence intensity based on IMU data alone. Using IMU data in place of a mounted anemometer could reduce payload and allow low-cost tracking of gas plumes, pollution, and weather patterns.
NP05.00048 Improving the Descent Performance of Small-Scale Rotorcraft through Added Geometries1. DANIEL YOS, MORTEZA GHARIB, MARCEL VEISMANN, Caltech — The descent stage of all rotor vehicles—from helicopters to drones—results in a significant loss of thrust and increased fluctuations with respect to the system in hover condition. These losses are believed to derive from the reijestion of the rotor flow that causes an accumulation of tip vortices at the rotor plane: often referred to as the vortex ring state (VRS). An approach of utilizing additional geometries within the proximity of the rotor plane was investigated by using enclosed shrouds and various props (with enhanced blade tip designs), in order to improve the stability and performance of rotorcraft in descent. These geometries were aimed to prevent the interaction between the blade tip vortices and the rotor disk. Results from single rotor thrust tests indicate that it is possible to reduce the thrust losses within the VRS by adding geometries in distinct locations relative to the rotor disk, while additional PIV analysis potentially outlines the underlying flow mechanism that causes these performance improvements.

1The presented material is based upon work supported by the Center for Autonomous Systems and Technologies (CAST) at the California Institute of Technology

NP05.00049 2D Particle Image Velocimetry and Computational Fluid Dynamics Study on Sidewall Brain aneurysm1. JACOB BARRERA, HAN HUNG YEH, DANA GRECOV, The University of British Columbia — Cerebral aneurysms are cerebrovascular abnormalities characterized by the weakening and dilatation of a localized cerebral arterial wall. They are a prevalent vascular disorder affecting 2–5% of the worldwide population. Aneurysm rupture can be fatal. To prevent aneurysm rupture, endovascular stents can be deployed to redirect blood flow away from the aneurysm, reducing blood flow velocity in the aneurysm sac and encouraging blood vessel remodeling. Since hemodynamics plays a key role in aneurysm progression, an in vitro experimental setup was developed to mimic the cerebral circulation with particle image velocimetry (PIV) testing different internal carotid artery sidewall aneurysm models. The Newtonian and the non-Newtonian working fluids with matching density and viscosity of human blood were used. In addition, a computational fluid dynamics (CFD) analysis was conducted in parallel. Computational model was first verified and validated against PIV measurements and showed good agreements. The study showed that the Newtonian model overestimated hemodynamic parameters, such as the blood velocity and wall shear stress in the cerebral aneurysm sac, comparing to the non-Newtonian model, suggesting the shear thinning effects might be more prominent in this region.

NP05.00050 Towards a better understanding of the flow mechanisms involved in blunt traumatic aortic rupture. GHASSAN MARAOUCH, CURTIS H. HORTON, EDUARDO MALORNI, JOSEPH FANABERIA, GIAN-CARLO MIGNACCA, LORENZO MERCURI-BASTIEN, LYES KADEM, Concordia University, MARK COHEN, PMG Technologies — Blunt traumatic aortic rupture is a heart injury that can occur in falls, automobile accidents, and sporting injuries involving impact to the thorax. Despite its severity and high morbidity rate, the research still does not provide a consistent description of the mechanism of rupture. In this study, a crash testing dummy with an in-vitro pumping heart, 3D printed ribcage, and ballistic gel damping layer was developed to reproduce a realistic response to thoracic impact. Testing was performed using a standardized pendulum used for calibration of crash test dummies, with the location of impact being the middle of the sternum. Different impact severities were tested by adjusting the kinetic energy at impact with the initial height of the pendulum. Measurements on the dummy include instantaneous aortic pressure waveforms during impact and accelerations at the spine and sternum. The results of this experiment show that aortic pressure experiences significant changes in magnitude during simulated impact. This work could help contribute towards a better understanding of the mechanisms leading to blunt traumatic aortic rupture and the development of preventative measures.

NP05.00051 ABSTRACT WITHDRAWN –

NP05.00052 ABSTRACT WITHDRAWN –

NP05.00053 Modelling flow driven by travelling wave motion in the glymphatic system. LOGAN BASHFORD, NOAH ANDERSON, AMY BURKE, JEFFREY TITHOF, DOUGLAS H. KELLEY, University of Rochester — The glymphatic system is a waste removal mechanism of the brain. The glymphatic pathway includes perivascular spaces (PVS), which are annular channels surrounding blood vessels in the brain. These channels are filled with cerebrospinal fluid (CSF). There is substantial experimental evidence that the pulsation of these blood vessels drives CSF flow through the PVS. Irregular or weak arterial pulsations may cause suboptimal flow and poor waste removal. A build-up of these wastes (e.g., amyloid-beta) is linked to the development of neurodegenerative disorders such as Alzheimer’s disease. This research introduces a laboratory model of an artery and surrounding PVS. A viscous fluid fills the annular space between a small flexible tube and a rigid transparent cylinder. A travelling wave is generated by moving spherical beads through the flexible tube. The speed of the travelling wave is varied, and flow between the cylinders is quantified with a Lagrangian particle tracking algorithm. It was observed that increasing the speed of the travelling wave increased flow speed in a linear manner. This work provides an experimental foundation to investigate the effect of other parameters, such as frequency, amplitude, and wave shape.

NP05.00054 Hydrodynamics of Lemon shark’s (Negaprion brevirostris) dorsal fins. VIVIAN TURNER, ROI GURKA, ERIN E. HACKETT, Coastal Carolina University — To improve understanding of the hydrodynamic functionality of dorsal fins of sharks, we focus on the Lemon shark (Negaprion brevirostris), which features a second dorsal fin that is almost as large as its first dorsal fin; an uncommon feature of migratory shark species that is not well understood from both a physical and biological perspective. Laboratory experiments are performed in a flume using PIV to measure the near wake flow behind the first and second dorsal fins, and behind the tail of a deceased Lemon shark as well as a 1:1 ratio 3D printed flexible shark model. Vortex shedding in the wake is characterized through POD applied to vorticity fields estimated from the PIV data. Hydrodynamic forces are estimated using the velocity deficit in the wake to estimate drag, and a thrust model based on the characteristics of the vortex street in the wake. Drag and thrust coefficient results behind the first dorsal, second dorsal, and caudal fin are compared.

NP05.00055 Thrust generation of fish-fin geometries in continuous rotation. EMMA JAMIN, CECILIA FUERTAS-CERDEIRA, MORTEZA GHARIB, California Institute of Technology — Fish are known to use flapping motions of their caudal fins to propel, while man-made propellers commonly use continuous motion to achieve thrust. It is not known if fish use the flapping due to its limited range of motion or if it is the most effective way to maneuver in the water. The objective of this work is to analyze the propulsive performance of propeller geometries similar to those of fish fins when performing continuous rotary motions. Because caudal fins possess distinctive morphologies and stiffness adapted to the fish specific modes of life, varying geometries and compliances have been considered. In order to evaluate the thrust generation properties of these fins, an underwater vehicle capable of generating continuous rotation and equipped with exchangeable propellers has been built and tested in a water tunnel. The thrust forces generated by the different fins are presented and compared.
NP05.00056 Vapor bubble condensation in Hele-Shaw cell submerged in subcooled pool, MASASHIRO OKADA, Division of Mechanical Engineering, School of Science and Technology, Tokyo University of Science, Japan, TAKUMA HORI, Department of Mechanical Systems Engineering, Tokyo University of Agriculture and Technology, Japan, ICHIRO UENO, Research Institute for Science and Technology, Tokyo University of Science, Japan — Cooling technologies based on boiling heat transfer have attracted attentions as the heat-generation density of electronic devices increases. It is known that the microbubble emission boiling (MEB) can overcome critical heat flux under the specific subcooled condition; thus, MEB regime is expected to be applied to next-generation cooling devices. However, the occurrence condition of MEB has not been yet understood. Since MEB has the remarkable feature that vapor bubbles abruptly condense and collapse, it is important to understand the condensation process of the vapor bubble. In order to focus on the condensation process of the vapor bubble, we inject vapor into a narrow gap region between two glass plates (Hele-Shaw cell) submerged in subcooled pool of water. The Hele-Shaw cell allows us to access clear visualization inside the vapor bubble. Special attention has been paid on the correlation between the condensation process of vapor bubble and energy exchange through the bubble surface.

NP05.00057 Development of a nanoscale hot-wire probe for supersonic flow applications¹, KATHERINE KOKMANIAN, Princeton University, SVEN SCHARNOWSKI, MATTHEW BROSS, CHRISTIAN J. KAÆHLER, Bundeswehr University Munich, MARCUS HULTMARK, Princeton University — A new sensor based on the nanoscale thermal anemometry probe (NSTAP) was designed and fabricated in Princeton University’s clean room to obtain well-resolved mass flux measurements in supersonic flows. In order to withstand high forces, the sensing element was redesigned to be 400 nm thick, 2 µm wide and 30 µm long. The sensor was tested in the Trisonic Wind Tunnel Munich (TWM) at Bundeswehr University. The TWM is a two-throat blowdown tunnel with a unique capability of altering both the Mach number and the Reynolds number independently and in real-time. Freestream measurements were taken at M=2 to investigate the convective heat transfer characteristics of the sensor. A linear calibration between the Nusselt number and the Reynolds number appeared to best fit the data. This linear Nu-Re dependence has previously been observed when operating hot-wires in free-stream flows. The sensor exhibits effective heat transfer (Nu-Re dependence on both width and thickness of sensing element), the sensor is believed to operate in slip flow conditions, exhibiting unique heat transfer characteristics. The importance of both the thickness and the width of the sensing element was also investigated theoretically for various Mach numbers.

NP05.00058 Three distinct liquid drop detachment dynamics on vibrating 1D rod structure, SSNG JUN LEE, KYEONGMIN KIM, WONJOON CHOI, Korea University — Along with physical impact of liquid drops, condensations can induce undesirable drop formations on solid surfaces. This outcome can possibly downgrade the heat transfer efficiency and can even contaminate sophisticated structures that may cause costs during microfabrication processes. This problem can be alleviated by vaporizing unwanted drops through controlling temperature or humidity. However, this method is cumbersome and cannot be applied in the case of viscous fluids. Thus here, we explore precise dynamics of overall fluid drop detachment on solid surfaces through damped harmonic oscillations. To model the complexity of the surfaces, we utilized 1D cantilever rods having high curvatures. The vibration of cantilever beams with small deflections (below 1 cm) were tested to see three different types of drop detachment behaviors depending on surface energy, fluid viscosities and volumes. Three dimensionless numbers (Weber, Capillary, and Bond numbers) were used to identify at which conditions the drops detach from the beam sides. Thus, not affecting neighboring structures or other parts of the same body. We concluded that high gravitational and inertial forces (Wₑ>10 and Ca/Bo<0.1) are favorable for clean drop detachment leaving no residue on solid surfaces.

NP05.00059 Generating Unsteady Pressure Gradients Using Rapidly Deforming Surface, AADHY PARTHASARATHY, THERESA SAXTON-FOX, University of Illinois at Urbana-Champaign — This study demonstrates the capability of generating unsteady pressure gradients by an experimental flow. Given the relatable nature of complex multi-stream supersonic nozzle, a high-speed camera was used to observe the dynamic pressure gradient, a car's rapidly deforming ceiling causes a dynamically strengthening favourable and adverse pressure gradient in sequence. Curvature of the ceiling is characterised in an instantaneous manner by using a high-speed camera, along with the corresponding spatial variation of the strength of FPG and APG imposed. The maximum speed of deformation of the ceiling is 1 m/s, and the acceleration parameter, k, at the section at the point of maximum deformation ranges between 2.5 x 10⁻⁸ and 8.5 x 10⁻⁹. Time-resolved planar Particle Image Velocimetry is used to investigate the behaviour of the flow over and around the deforming ceiling. The different geometric and dynamic conditions tested will allow investigation of the effect of dynamic pressure gradients on the behaviour of turbulent boundary layers.

NP05.00060 Comparison of Complex Multi-Stream Supersonic Nozzle Geometry¹, TYLER VARTABEDIAN, SETH KELLY, EMMA GIST, DOMINIC DIDOMINIC, MARK GLAUSER, Syracuse University — Noise continues to be a concern with further developments of high-speed flow and the structural geometries surrounding it. A rectangular multi-stream supersonic nozzle with an aft deck is resolved utilizing stereo PIV along with near and far-field pressure and acoustic measurements. A focus is placed on optimizing for noise reduction while altering aft deck geometry through the development of a previously trained neural network. This is accompanied by incorporating a varied splitter plate which decomposes the flow field into two canonical flows, a core supersonic flow interacting with a subsonic wall-jet. The synergy of these two flows creates complex turbulent structures which feed into amplifying noise. Through the combined data from stereo PIV, near and far-field pressure and acoustic measurements, the goal is to design a low noise aft deck plate while furthering an understanding of how the splitter plate geometry affects the multi-stream flow interaction and relevant acoustic measurements.

¹This study was provided funding via research grants by the Air Force Office of Scientific Research (AFOSR) grant number FA9550-19-1-0081 (Program Manager: Dr. Greg Abate) and Spectral Energies LLC.

NP05.00061 Analysis of an acoustic levitator flow field using Particle Shadow Velocimetry, AMY LEBANOFF, Karlsruhe Institute of Technology, German Academic Exchange Service Research Internships in Science and Engineering, University of Central Florida, KAI SCHOENEWOLF, STEPHAN AUTENRIETH, CHRISTIAN LIEBER, RAINER KOCH, HANS-JOERG BAUER, Karlsruhe Institute of Technology — High concentrations of Nitrogen Oxide (NOx) emissions from diesel engines pose a threat to the environment. However, selective catalytic reduction (SCR) offers a means to reduce these NOx emissions. Optimization of SCR technology requires study of urea-water-solution (UWS) evaporation behavior to tune droplet evaporation models. Validation data for this purpose may be obtained via observation of acoustically levitated droplets under controlled pressure, temperature, humidity, and flow field conditions. Establishing well-defined conditions prior to actual evaporation experiments is vital for model development. One driving factor behind the heat and mass transfer associated with droplet evaporation is the relative velocity imposed by the flow field of the gas phase near the droplet. To characterize the levitator flow field, Particle Shadow Velocimetry was employed. The optical setup features a double-frame camera equipped with a long-distance microscope that is maneuverable along three axes, allowing for targeted assessment of locations in the acoustic levitator without adjusting the illumination. This measurement technique facilitated incremental improvements which resulted in a symmetric flow field deemed suitable for investigation of UWS droplet evaporation.
NP05.00062 Drive an Object Using Photothermal Convection Around a Water Vapor Microbubble. RYUTA MATSUMURA, SOUKI IMAFUKU, KYOKO NAMURA, MOTOFUMI SUZUKI, Kyoto University, MICRO PROCESS ENGINEERING LABORATORY TEAM — By using the photothermal heating of the thin film under laser irradiation, a water vapor microbubble is formed in degassed water. The bubble involves rapid flow, which results from the Marangoni force and bubble oscillation caused by a steep temperature gradient. It is assumed that the flow direction is changed by giving an asymmetric temperature gradient. Therefore, we changed temperature distribution by laser irradiation with multiple spots. By focusing a laser spot on the thin film immersed in degassed water, a water vapor bubble with a diameter of approximately 10 µm was created. Simultaneously, a sub laser spot was focused next to the bubble to yield a temperature gradient in the direction parallel to the film surface. Consequently, the rapid flow was generated around the bubble, whose direction was dependent on the power and position of the sub. Then, we formed on a thin mica chip as a lighter substrate the bubble which can generate flow parallel to the film surface. Finally, we succeeded in moving the mica chip by using the reaction force of the photothermal convection. It is expected to be utilized as the technique for a driving force in microfluidics.

1This project has received funding from JSPS KAKENHI Grant Nos 17H04904 and 17H01050.

NP05.00063 Double-diffusive and Rayleigh-Taylor instabilities in particle-laden water stratified over salt water in a Hele-Shaw cell. GRAHAM CHAMBERS-WALL, CATHERINE DEMA, ECKART MEIBURG, Department of Mechanical Engineering, University of California at Santa Barbara, Santa Barbara, CA 93106 U.S.A., PATRICK BUNTON, Department of Physics and Mathematics, William Jewell College, Liberty, MO 64068 U.S.A. — An experimental and computational investigation is performed for double-diffusive (DD) and Rayleigh-Taylor (RT) instabilities in particle-laden fresh water initially stably-stratified over salt water in a Hele-Shaw cell. Computationally, Darcy’s Law coupled with an advection-diffusion equation for salt and an advection-diffusion equation for particle concentration that includes a settling velocity is solved for two-dimensional stratified fluids in the presence of particle-loading. The flows are parametrized in terms of a stability ratio, a gravity parameter, and a dimensionless settling velocity. Results are analyzed in terms of relative dimensions of concentration profiles of sediment and salt. Experimentally, Schlieren imaging is used to image fresh water containing 3-6 µm glass microparticles layered above salt water both containing glycerol to slow dynamics. Dimensionless wavelength and time and distance until onset of instability are measured. Results are interpreted in terms of a “nose” region of increased density at the interface.

1This research was supported by NSF grant CBET-1914797.

NP05.00064 Experimental analysis of dilute particle-laden liquids over and through patterned structures. EILEEN HAFFNER, University of Illinois at Chicago, JONATHAN HIGHAM, University of Liverpool, PARISA MIRBOD, University of Illinois at Chicago — Particle-laden liquids are encountered in various applications both in laminar and turbulent flows. However, the concentration and velocity profiles of dilute suspensions when they are flowing over and inside a patterned surface are not yet known. This experimental study is conducted to examine the interaction of the particles in various dilute suspensions over and through a patterned structure. The patterned surface consists of cylindrical rods arranged in a square array. Particle Tracking Velocimetry data provides velocity and shear rate at the interface between the free flow region and the surface for dilute suspension flows. We examined the velocity and concentration profiles through and above the structure for various dilute suspensions. We find that the shear rate and velocity profiles are strongly dependent on the suspension properties and geometry of the structure.

1National Science Foundation award 1854376 and Army Research Office award W911NF-18-1-0356

NP05.00065 Accumulation structure of low-Stokes-number particles in high-aspect ratio half-zone liquid bridge of high Prandtl number fluid. TOMOKI SAKATA, HIROKI SAITO, Division of Mechanical Engineering, School of Science and Technology, Tokyo University of Science, Japan, TAKUMA HORI, Department of Mechanical Systems Engineering, Tokyo University of Agriculture and Technology, Japan, ICHIRO UENO, Research Institute for Science and Technology, Tokyo University of Science, Japan — Various space experiments on the International Space Station (ISS) have been conducted to elucidate the Marangoni convection in liquid bridge formed under the microgravity environment. Since the cost of space experiments is apparently expensive, it is necessary to accumulate knowledge of the phenomena on ground as much as possible in term of preliminary experiments. In addition, ground experiments can allow us to clarify the effects of gravity. We focus on the particle accumulation structure (PAS) produced by the thermocapillary effect in a half-zone liquid bridge on the ground. To compare with the PAS realized in Japanese Experimental Module Kibo in the ISS, the PAS in high-aspect ratio half-zone liquid bridge of high Prandtl number fluid is studied. The role of gravity is discussed through the comparison of these experimental results also together with linear stability analysis.

NP05.00066 Water entry of hydrophilic spheres through fabric-fluid interfaces. DAREN WATSON, CHRIS SOUCHIK, JOSHUA BOM, ANDREW DICKERSON, University of Central Florida — The vertical impacts of solid projectiles with the free surface of a deep aqueous pool are traditionally investigated with respect to impactor shape, entry speed, and surface roughness. Free surface alteration in some cases, may be more readily achieved for the modulation of splashes. In this combined experimental and theoretical study, smooth, free-falling, hydrophilic spheres impinge on penetrable and non-penetrable fabric substrates resting atop the fluid surface for Weber numbers in the range of 430-2700. Penetrated fabrics remain near the free surface, suppressing the splash crown, but allowing passage of a Worthington jet whose height increases with the depth of the trailing cavity. Non-penetrable fabrics create deep seal cavities by veiling the descending impactor, generating higher Worthington jets, and pronounced splash crowns. Some fabrics, both penetrable and non-penetrable reduce drag with respect to clean surface impacts by providing the drag-reducing benefits of flow separation while not offering a high inertial penalty. Such observations augur well for military and industrial applications where splashless warrant control to mitigate damage to life and property.

NP05.00067 Plume studies to characterize Turbulent Buoyant Plumes using multiple sensors. DANIEL BRUN, SUDHEER REDDY BHIMIREDDY, KIRAN BHAGANAGAR, University of Texas at San Antonio - Turbulent axisymmetric buoyant plumes released into calm air are studied experimentally using multiple sensing techniques by a hot-wire anemometer, Schlieren imaging and a FLIR VUE pro thermal camera. Heated Carbon Dioxide is released into a plume chamber under controlled conditions at a nozzle exit Reynolds number ranging from 1300 to 2000. The buoyancy flux and momentum flux at the nozzle exit are determined from the flow behavior as a function of initial conditions. Time-averaged statistics such as centerline velocity, temperature and plume half-width are calculated using hot-wire readings and image-processing of thermal camera and Schlieren recordings. To better understand the effect of buoyancy on turbulence and mean velocity, a reference case with no buoyancy flux at nozzle exit is studied.
NP05.00068 Interactions between bathtub vortices in a rotating experiment¹, DANIEL VAN BEVEREN, Haverford College. DANIEL LATHROP, University of Maryland — The bathtub vortex is a flow pattern uniquely suited for laboratory study, as the flow through a drain hole provides a radial inflow and vortex stretching in a controlled location, thus stabilizing the vortex. While these radial flow patterns provide stability, the attraction they cause between any number of such vortices makes it difficult to achieve a stable state of multiple bathtub vortices, thus limiting their utility as models for experimental study of vortex interactions. Here we test the effect of global rotation on such vortices by spinning a cylindrical container with two drain holes in the bottom, which we find allows the formation of multiple bathtub vortices in a co-rotating stable state. The orbit period of these vortices is observed to change with the global rotation rate, and apparently spontaneous switching between states with different numbers of vortices is observed, with larger numbers of vortices possible at higher global rotation rates.

¹ NSF Award PHY-1756179 and EAR-1909055

NP05.00069 Flow induced vibrations from two circular cylinders in close proximity, CHRISTOPHER O’NEILL, APRIL JANG, BRANDON MCNEELY, ROBERT MARTINUZZI, CHRIS MORTON, University of Calgary — The present study is focused on investigating flow induced vibrations (FIV) of two circular cylinders (of diameter D and D/8) in proximity. The smaller diameter cylinder (referred to as the ‘control’ cylinder) is controlled via a two degree of freedom (x,y) traverse system. The system uses a genetic algorithm in an effort to maximize the amplitude response of the FIV of the larger diameter cylinder (referred to as the ‘main’ cylinder). Due to the difference in cylinder diameters and corresponding natural frequencies, at flow velocities where FIV begins to occur for the main cylinder, the control cylinder vibrates with a significant amplitude. To combat this issue, passive flow control methods are employed (i.e., helical strokes). The amplitude response of the main cylinder is investigated in detail in the present study using a combination of time-resolved planar FIV measurements and displacement measurements. The results show that the amplitude response and corresponding wake dynamics of the main cylinder are impacted significantly by the control rod and its positioning relative to the main cylinder.

NP05.00070 Noise Characterization of a Two Circular Cylinder Flow Induced Vibration Energy Harvester System, APRIL JANG, CHRISTOPHER O’NEILL, BRANDON MCNEELY, ROBERT MARTINUZZI, CHRIS MORTON, University of Calgary — The development of flow induced vibration (FIV) energy harvesters is an active area of interest as an alternative to traditional hydropower systems. We have developed a mechatronic system to investigate FIV of two circular cylinders (of diameter D and D/8) in proximity. The smaller diameter cylinder (referred to as the ‘control’ cylinder) is controlled via a two degree of freedom (x,y) traverse system. The system uses a genetic algorithm to find the optimal parameters describing a sinusoidal motion of the control cylinder, in order to maximize the amplitude response of the FIV of the larger diameter cylinder. In testing this system, various sources of noise have been identified that disrupt the genetic algorithm’s ability to find the optimal control cylinder parameters. To minimize the impact of noise on the system, a secondary genetic algorithm will be used to characterize the noise properties of the system as a function of the control cylinder parameters. Insights from this analysis will allow for modifications to both the mechanical system and the software to improve the overall performance of the system.

NP05.00071 Controlling chaos by the domain size, MAHDI GHADIRI MOTLAGH, ROUSLAN KRECHET-NIKOV, University of Alberta — As part of the recent effort to understand dynamics and evolution on time-dependent spatial domains, we present an experimental investigation on how domain deformation may serve as a mechanism regularizing chaotic motion. Faraday waves – standing waves formed on the free surface of a liquid layer due to its vertical vibration – are chosen here as a paradigm owing to their historical use in testing new theories and ideas. In our experimental setup of a vibrating water container with controlled positions of lateral walls, the Faraday patterns are visualized using the Fourier transform profilometry and the wave amplitude is measured using a high accuracy laser displacement sensor: these techniques allow us to reconstruct a time history of the pattern three-dimensional landscape. Data analysis reveals that domain deformation is not only able to transform the chaotic state of two competing modes into a regular (periodic) one, but also to isolate one of the competing modes in the regime, which on a time-fixed domain of the same size would otherwise correspond to a regular or chaotic pattern competition. These experimental findings are interpreted with appropriate theoretical arguments and insights.

NP05.00072 Measurements of Reacting Fuel Sprays Using High-Speed Imaging¹, JACKLYN HIGGS, JOSHUA BITTLE, The University of Alabama — Engineers are currently looking for solutions to reduce waste and the effects transportation needs have on the environment. One method is to transition to biofuels in diesel engines. Computational Fluid Dynamics (CFD) simulations can be useful provided enough experimental data is available for validation. Hence, we need real experiments to determine the relationship between simple properties of fuels that we can measure and the results of actual combustion experiments. An optically accessible constant pressure flow rig (CPFR) is the primary experimental apparatus and it can be used to set control parameters to study the fuel-air mixing in conditions similar to diesel engines. A schlieren camera measures initial fuel-air mixing, a chemiluminescence camera measures initial combustion, and a two color pyrometry camera measures soot production. Extensive test campaigns at various injection conditions, ambient conditions, and fuel type will enable a new level of understanding of the diesel combustion process. Before significant testing can be completed, the primary focus of the work presented here is on efforts to optimize the experimental set-up by addressing some key experimental challenges that had previously limited the quality of data obtained. Significant effort was also dedicated to developing code to aid in processing a limited initial data set. This code serves as a proof of concept that can be leveraged for larger data sets to be acquired in the future. As a result, the lab is an efficient and effective work space that allows for ease with acquiring data.

¹ NSF Grant Number 1659710

NP05.00073 Predicting Plasma Plumes Induced by High-Powered Lasers on a Copper Plate under Different Conditions¹, REECE FREDERICK, None — Industries that require material removal on a micro level, e.g., microelectronic industry, utilize high powered lasers that essentially vaporize the material away. When such high-power lasers strike materials, however, they often create plasma. The plasma plume absorbs the laser radiation before it reaches the target, reducing the rate of material removal. In the present work, plasma plumes induced by irradiation of a copper plate with a short-pulse laser under various background gas pressures are studied experimentally. The plasma is recorded using Schlieren imaging and a highspeed camera. This project is aimed to recreate the plasma environment in order to better predict its behavior. With better understanding of the plasma, we can more efficiently use these lasers to remove material by minimizing plasma absorption effects.

¹ OIA-1655280
NP05.00074 Preliminary Characterization of an Iodine Plasma Source for use in Material Analysis

GEORGIA SHARP, JAMES ROGERS, RICHARD BRANAM, University of Alabama — Electric propulsion devices using xenon as a propellant are a high efficiency solution for large conventional satellites. The high storage density of iodine would enable these devices to require less mass for use in space technologies, if used as a propellant as an alternative to xenon. The ability to reduce the mass required for electric propulsion devices would not only reduce costs of space travel but also open up new opportunities for these devices to be used in smaller, more volume constrained missions. Iodine is a strong oxidizing agent. To determine if it is a viable alternative, the erosive properties must be quantified. The object of this research project was to characterize an iodine plasma source before using it for material exposure and analysis. A double Langmuir probe was used as the method of data acquisition for the plasma conditions. The plasma characterization identified the conditions in the plasma source that will be used to properly quantify the erosive properties in iodine plasma. Preliminary results indicate a maximum electron temperature of four electron volts, and a maximum plasma density of eight inverse cubic centimeters.

1This work is supported by the NSF Grant #1650710, the NSF EPSCoR RI-Track-I Cooperative Agreement OIA-1655280, and the Air Force Office of Scientific Research grant FA9550-17-1-0204.

NP05.00075 Measuring Pressure and Strain with Luminescent Coatings

KIMBERLY LOWNDES, Berry College, KYLE CHISM, AMRUTHKIRAN HEGDE, JAMES HUBNER, The University of Alabama — Often, researchers employ probes such as pressure taps and strain gauges to measure the pressure and strain on aerodynamic objects. However, these tools lack high-resolution and full-field capabilities that may be necessary for high-speed aerodynamic testing. A combination of photoelastic coatings (PEC) and pressure sensitive paint (PSP) has the potential to provide researchers with correlated, full-field surface measurements of maximum shear strain and pressure, respectively. Photoelastic coatings use circular polarized light along with birefringent material properties to provide information about the surface strain of objects, while pressure sensitive paint utilizes oxygen-quenched luminophores to measure pressure. Benchtop test results will be presented of a dual-layer PEC/PSP coating applied to cantilever specimens subjected to static and dynamic loading and imaged with a micro-polarizer digital camera.

1NSF REU Grant: EEC 1650710 and AFOSR Grant: CBET-1802994

NP05.00076 Can we measure the three-dimensional orientation of Vorticella from two-dimensional videos?

LUKAS KAROLY, RACHEL PEPPER, University of Puget Sound — Vorticella are aquatic suspension feeding microorganisms that live attached to surfaces and generate a feeding flow to draw in their food. They are crucial players in aquatic ecosystems, eating bacteria and debris as well as supporting larger aquatic organisms. To evaluate the impact of Vorticella feeding microorganisms that live attached to surfaces and generate a feeding flow to draw in their food. They are crucial players in aquatic ecosystems, eating bacteria and debris as well as supporting larger aquatic organisms. To evaluate the impact of Vorticella in their environments, as well as in practical applications like waste water treatment, it is important to understand Vorticella feeding rates. Previous work has shown that the orientation of Vorticella relative to the surface of attachment affects feeding flow and feeding rates. Vorticella cell body orientation is defined by the polar angle, which is measured from the vertical axis, and the azimuthal angle. Previous experiments have observed Vorticella using a horizontal microscope from which the polar angle was directly measured. The azimuthal angle was inferred as a function of the projected cell body length compared to a maximum measured cell body length. However, it is unknown how accurately this technique determines the azimuthal angle. We recorded time-lapse videos of Vorticella simultaneously from the side and the top. We then compared the calculated azimuthal angle from the side view to a direct measurement from the top view. We report the error in the calculated azimuthal angle as a function of the organism orientation.

NP05.00077 Influence of characteristic components of blood on blood splashing onto a solid wall

YUTO YOKOYAMA, MASANORI TAKEDA, HAJIME ONUKI, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology — Understanding of blood splashing on a solid wall is of great importance in forensic science since blood splashing determines the blood pattern remained on the wall. Blood is known as a complex liquid, containing platelets and blood cells such as red blood cells and white blood cells, as well as coagulant factors and liquid components such as water and proteins. In this study we investigate influence of characteristic components of blood on its splashing onto a solid wall by separating main components of the bloods. In our experiment, whole blood, containing all of the components, can be separated into three types of liquids: (i) Serum containing only liquid components of blood, (ii) PRP containing serum with coagulant factors, (iii) Plasma containing PRP with platelets. The droplets of these liquids and blood simulants are dropped from a needle at various heights and recorded by a high-speed camera. It is found that the droplet of whole blood shows a quite different behavior from droplets of the other types of liquids. We also compare the experimental results with the theories proposed recently and discuss the effect of the blood components on splashing.

1JSPS KAKENHI Grants (No. 17H01246 and 17H02808)

NP05.00078 Wind tunnel testing of a NACA 0012 airfoil with passive biomimetic flow control devices

CHRIS JARMON, AMY LANG, PAUL HUBNER, SEAN DEVEY, The University of Alabama — Increased demand for eco-friendly and energy-efficient transportation has led researchers to explore methods of drag reduction for decades. Previous studies have shown some success by both passive and active flow control techniques. Recent studies have shown that shark skin has the ability to passively alter the flow by inhibiting flow reversal and controlling flow separation. This study examines various 3D printed biomimetic flaps and scales inspired by sharks and birds to begin to determine optimum design parameters such as the presence of surface riblets and characteristic length. The biomimetic surfaces are placed on the upper surface of a NACA 0012 wing. Force data is acquired for baseline and biomimetic surface models (Re = 200,000) at fixed angles of attack (steady case), gradual pitching rate (quasi-steady case), and high frequency pitching rate (dynamic case). Results will be presented as to the correlation between the biomimetic flaps and the alteration of airfoil lift and drag.

1Funding from US Army grant W911NF1510556 and NSF REU grant EEC 1650710
NP05.00079 Particle alignment under unsteady shear in non-Newtonian fluids causing modulations of effective viscoelasticity1, TAIKI YOSHIDA, Graduate School of Engineering, Hokkaido University, YUJI TASAKA, YUICHI MURAI, Faculty of Engineering, Hokkaido University — The effective viscoelasticity modulated by particle alignment in unsteady shear in non-Newtonian fluids was revealed by means of ultrasonic spinning rheometry (USR) [Yoshida et al., J. Rheol.(2019)]. The dispersed particles make alignments in the sheared direction in unsteady shear flows when the fluid media have sufficiently long relaxation time. USR evaluated the effective rheological properties modulated by the particle alignments; (1) the effective viscosity does not reach the value estimated by Einstein’s law; (2) the effective elasticity increases significantly with the increasing volume fraction of particles in the bulk of the measurement volume. To clarify factors causing the particle alignment in unsteady shear flows/deformations, numerical tests using a simple toy model with considering spring forces connecting between the particles with specific yield stresses were examined. The numerical tests explained the importance of the relaxation process on the orientation of the particles. To conclude the experimental findings supplemented with the results of the numerical tests, we suggest that local and macro rheological characteristics are strongly modulated by the particle alignment when the test fluid media have long relaxation times.

1This work was partially supported by a Grant-in-Aid from the Japan Society for the Promotion of Science (JSPS) Fellows, No. 18J20516, also by JSPS KAKENHI No. 19H02057.

NP05.00080 Direct and indirect influence of crystallization on double-diffusive convection in ammonium-chloride solutions, DAISUKE NOTO, Hokkaido University, STEN ANDERS, SVEN ECKERT, Helmholtz-Zentrum Dresden-Rossendorf, YUJI TASAKA, Hokkaido University — Crystalization inside fluid flow is a complicated, but highly fascinating phenomenon in the field of geo- or astrophysics. For instance, a magnetic field of Jupiter’s moon Ganymede is thought to be sustained by crystalizing flow termed iron-snow, but little is known up to date. Thus, experimental tests to seek likely conditions for a presence of a dynamo are required. We have approached these phenomena via a model experiment using a test fluid of aqueous ammonium-chloride solution, which changes phase from liquid to solid under room temperature. To quantify velocity and temperature fields simultaneously, thermochromic liquid crystals (TLC) were suspended into the test fluid. We have established a masking technique to obtain velocity fields of the continuous and the solid phase. In addition, a neural network based thermometry utilizing TLC coloration has been established. With these methods, we found a direct and indirect influence of the crystals on the flow. At the beginning of the cooling process, intense precipitation of equiaxed crystals can directly modify the flow structure. Meanwhile, columnar crystals start to grow, and impede the cooling from the wall. Indirectly, crystal growth creates a stable density stratification, but an unstable temperature stratification.

NP05.00081 Experimental study on flow-induced vibration of tandem flexible cylinders at varying angles of inclination, GIANCARLOS CASTRO CASTRO, Miami University, BANAFSHEH SEYED-AGHAZADEH, University of Massachusetts, Dartmouth — Flow-induced vibration (FIV) response of a highly flexible inclined circular cylinder placed in the wake of a stationary cylinder is studied experimentally. The flexible cylinder is tension-dominated with an aspect ratio of 47 and a high mass ratio of 120. The cylinder was held fixed at both ends and placed in the test-section of a subsonic wind tunnel. The angles of inclination were varied from 0 to 45° with increments of 15. The inclined flexible cylinder lied in the wake of an upstream stationary cylinder of equal diameter and inclination. The dynamic response of the downstream flexible cylinder is studied for center-to-center spacing range from 3 to 7 times the cylinder diameter, in the reduced velocity range of $Re=260-3750$. Influence of inclination and cylinder spacing is investigated through studying the dynamic response of the cylinder in terms of the excited structural modes, amplitudes and frequencies of oscillations and transition between modes. Different values observed for the onset of oscillations and modal weight contributions explains that the FIV response of the system is different from that of a completely vertical cylinder for all angles of inclination larger than 15°. Dynamic response of the flexible cylinder was found to be always under the influence of the upstream one, even for large cylinder spacing of 7d.

NP05.00082 ABSTRACT WITHDRAWN

NP05.00083 ABSTRACT WITHDRAWN

NP05.00084 Effect of Seed Density on Dispersal of Seeds from Wet Splash Cup Plants, KELSY BRYSON, EMILY SAWICKY, RACHEL PEPPER, University of Puget Sound — Splash cup plants use raindrops to disperse their seeds. Plants are approximately 4-30cm tall with ~5mm-diameter fruit bodies. When raindrops fall into their conical fruit bodies, the splash ejects the seeds up to 1 m away from the parent plant. Understanding how the seeds are projected may enable a deeper understanding of dispersal after drop impact in other contexts and of splash cup plant evolution. Previous work using 3D printed cones as fruit body mimics found that maximum dispersal occurs with a 40° cone angle, defined as the angle between the side of the cone and the horizontal. Later work found that seeds, which were not accounted for in the original study, decrease dispersal distance. Seed density has also been found to correlate inversely with the average dispersal distance of seeds projected from a dry cup. In this study, we investigate the effect of seed density on dispersal from a cup that contains both seeds and water since this situation is commonly found in nature after one splash has occurred. We use 3D printed cones, a low-density polyethylene seed mimic, a high-density glass seed mimic, and high-speed video to analyze the splash. Our results show that lighter seeds travel further than heavier seeds, and the optimal cone angle remains 40° for both densities.

NP05.00085 Inertial, Aerodynamic and Elastic scaling of a passively pitching insect wing1, KIT SUM WU, JEROME NOWAK, KENNETH BREUER, Brown University — Evidence suggests that prominent features of insect wing pitching behavior are affected by inertial and aerodynamic forces with largely passive contributions from the wing hinge joint, which acts like a torsional spring. Motivated by the robotic applications of insect-inspired passive-pitching flapping wings, we study the scaling relationship between aerodynamics, inertia, and elasticity in the regulation of wing pitch and in the generation of lift forces in hovering flight. We measure forces and wing kinematics using an under-actuated robotic model with a prescribed wing stroke and an elastic wing hinge. Our data show that the fundamental dimensionless parameters for wings of varying geometrical, physical, and operational parameters. We also observe a dependency of pitching kinematics on these dimensionless parameters, providing a connection between lift coefficient and pitch angle characteristics. Our results illustrate the trade-off between contributions to lift by quasi-steady and rotational dynamics associated with wing translation and wing rotation at stroke reversal. These results will be of value both in understanding the mechanics of insect flight as well as in the future design of under-actuated flapping aerial robots.

1Supported by NSF
NP05.00086 Waterbowls: Reducing Impacting Droplet Interactions by Momentum Redirection. HENRI-LOUIS GIRARD, DAN SOTO, KRIPA VARANASI, MIT. — Droplets impacting a solid surface can transfer mass and energy to that substrate. While superhydrophobic surfaces can restrict this transport by making drops bounce off, the liquid-solid contact is still extensive. Recent studies aiming to limit this transport further have focused on reducing the contact time. Here, we remark that flux-based transport phenomena scale with the contact area as well as time leading us to define an Interaction Parameter as the integral of the contact area as a function of time to describe the drop-substrate interaction. We design superhydrophobic surfaces with a macroscopic structure that redirects the momentum of the spreading lamella upwards, thereby restricting the liquid-solid contact. We show that, with optimally designed, such surfaces can reduce the interaction parameter by an order of magnitude compared to a regular superhydrophobic surface and provide design guidelines for the macroscopic structure. Finally, we demonstrate that a well-designed surface can reduce heat transfer between a simulated rain and a solid surface by a factor of two.


NP05.00088 Analysis of Shear Layer Structures from Time Resolved Schlieren Images of Supersonic Multi-Stream Rectangular Nozzle Flow. CHRISTOPHER HAUCK, JACQUES LEWALLE, MARK GLAUSER, Department of Mechanical and Aerospace Engineering, Syracuse University, Syracuse, New York. — Operational guidelines for new jet engines result in extreme flow physics. Using a Single-Expansion-Ramp-Nozzle (SERN), Syracuse University and The Ohio State University study the effects of variable operating conditions, nozzle configurations, and deck plate geometries on jet development and far-field acoustics. Identifying noise sources will allow the implementation of control systems to reduce the overall sound pressure levels. Building on previous APS presentations, Ruscher 2015 and Tenney 2018, this project uses time resolved (100 and 400 kHz) Schlieren imaging. Through filtering, these Schlieren images show dark “blobs” convecting along the top shear layer, and seemingly synchronized “bands” propagating off the exit shock wave. Upon analysis, these flow structures occur at a frequency of 34kHz. The frequency of the phenomena are due to von-Karman streets from the third-stream splitter plate, triggering Kelvin-Helmholtz Instabilities [KHI] in the shear layers. Analysis is occurring on the relationship between the KHI occurrences and resulting aeroacoustics. Ongoing work includes processing alternate image orientations and new data acquisition for variable splitter and deck plates.

1This study was provided funding via research grants by the Air Force Office of Scientific Research (AFOSR) grant number FA9550-19-1-0081 (Program Manager: Dr. Greg Abate), Spectral Energies LLC, and the Undergraduate Research Program at Syracuse University.

NP05.00089 Using Traveling Water Beads for Particle or Vapor Capturing. ABOLFAZL SADEGHPOUR, Mechanical and Aerospace Engineering Department, University of California, Los Angeles, HANGJIE LI, Department of Mathematics, University of California, Los Angeles. Y. SUNGTEAK JU, Mechanical and Aerospace Engineering Department, University of California, Los Angeles, ANDREA L. BERTOZZI, Department of Mathematics and Mechanical and Aerospace Engineering Department, University of California, Los Angeles. — Here, we report a new dehumidifier design with 200% higher water condensation rate per volume of the device compared to the traditional dehumidifiers while decreasing air-side pressure drop due to providing straight gas flow channels for the gas stream. This structure consists of cold fresh water beads traveling down along an array of vertically aligned strings, on which the counterflowing hot humidified air stream is condensed. The water beads are generated using the intrinsic instability of liquid films flowing down a vertical thread. Offering direct contact between the water and air stream, very high interface-to-volume ratios and long resistance times for heat/mass transfer, our unique dehumidifier is capable of achieving superior heat and mass transfer rates. Additionally, using our bead generating method, we developed an efficient particle capturing unit, in which an array of traveling beads is used as the collection section to capture charged ultra-fine particles (UFP) in the countercflowing air stream. Studies show that UFPs can enter the blood and penetrate the cell membranes. Our experimental results show that this design can remove more UFPs with significantly lower water consumption rate compared to the traditional particle capturing systems.

1U.S. NSF grant CBET-1358034 and Simons Foundation Math+X Investigator Award 510776.

NP05.00090 Plasma Assisted LCF Perovskite Ion Transport Membrane Processing of Methane. JULIO OCANA ORTIZ, CHIGOZIE CHINAKWE, REECE FREDERICK, RAJAGOPALAN RANGANATHAN, MRUTHUNJAYA UDDI, None — Ion Transport Membranes (ITM) are ceramic based materials that allow the permeation of ions, commonly oxygen, through the structure at high temperatures. ITMs can be utilized in oxy-combustion, molecular separation of hydrogen reactors, and is effective in Carbon Capture and Sequestration (CCS) methods. Nevertheless, such high operating temperatures are not completely ideal, mainly due to its environmental impact and high energy input. In this project, it is aimed to reduce the temperature required to enhance oxy combustion reactions by applying a low-temperature plasma-catalysis to a LaCaFeO3 (LCF) perovskite ITM. The experimental reactor is placed inside a ceramic casted furnace where the reaction takes place on each side of the membrane, monitoring the products on the sweep side using a Quad-Pole Mass Spectrometer (QPM). Future works aim to use solar energy to power the reactor.

1NSF EPSCOR Research Infrastructure Improvement (RII) Track-1 Cooperative Agreement OIA-1655280 grant and administered through the CPU2AL: Alabama Plasma Internship Program at the University of Alabama.
NP05.00091 Plasma Assisted LCF Perovskite Ion Transport Membrane Processing of Methane.\textsuperscript{1} CHIGOZE CHINAKWE, JULIO OCANA-ORTIZ, None, REECE FREDERICK, R. RANGANATHAN, DR. MRUTHUN-JAYA UDDI TEAM — Ion transport membranes (ITMs) have played a key role in the study of processing and converting natural gases. The membranes are composed of ceramic-based materials and allow the permeation of ions at temperatures ranging from 700° to 950°. Due to low energy conversion efficiency, the high temperatures related to the membrane separation system are inefficient for industry use. The experiment at hand focuses on treating LaCaFeO\textsubscript{3} (LCF) perovskite ITM with low-temperature plasma in order to lower the activation energy and heighten the performance of the ITM. The reactor was controlled in a two-inch diameter quartz tube placed inside an alumina ceramic furnace. Metal inlets sealed the opening ends of the quartz tube and allowed the insertion of the LCF perovskite ITM, feedstock gas (air), sweep gas or reactive gas (CH\textsubscript{4}), and plasma electrode (kanthal wire). To generate the plasma, the electrodes were connected to a 110 V PVM/DDR plasma drive. The study was inconclusive due to the damaging of the LaCaFeO\textsubscript{3} perovskite ITM. The bulk of the ITM, as well as the adhesive used to attach the membrane, assisted in the ITM rupturing. Moving forward we hope to find a solution to these problems in order to run the reactor and receive data.

\textsuperscript{1} NSF EPSCOR Research Infrastructure Improvement (RII) Track-1 Cooperative Agreement OIA-1655280 grant

NP05.00092 Particle enrichment and instability on a fluid interface\textsuperscript{1} BENJAMIN DRUECKE, ALIREZA HOOSHANGINEJAD, University of Minnesota, JENNA BROWN, Fort Lewis College, SUNGYON LEE, University of Minnesota — We investigate the displacement of a suspension of non-colloidal particles by an immiscible fluid inside a vertical Hele-Shaw cell with a gap less than twice the nominal particle size. We find that the particles move slower than the invading fluid and accumulate on the interface. The particle enrichment can cause an interfacial instability reminiscent of the classic Saffman-Taylor instability. However, unlike the classic viscous fingering patterns, the invading fluid penetrates into regions surrounding clusters of high particle concentration. In this way, the high-concentration clusters deform the otherwise flat interface. Although this effect is enhanced by the presence of many particles in a cluster, we show that the instability can also occur in the case of a single particle for a narrower range of parameters. In this poster, we present experimental results and discuss the competition between viscous drag and interfacial energy giving rise to this instability.

\textsuperscript{1} Funded by the National Science Foundation through the University of Minnesota MRSEC under Award Number DMR-1420013

NP05.00093 Flexible Airfoils and Their Effect on Flow Separation\textsuperscript{1} DAVID FARIYIKE, The University of Alabama, KELLIS KINCAID TEAM, LALIT ROY TEAM, DR. DAVID MACPHEE TEAM, ROOIHANY MAHMOOD TEAM — In the US alone there are 5,000 planes in flight at any given moment and 52,000 wind turbines in operation. Any object that is subject to high wind speeds or varying attack angles has the potential to have flow separation. Flow separation increases drag which results in a less efficient aerodynamic system. Previous research has shown that active shape changing airfoils can reduce flow separation. However, since the shape change is active it introduces parasitic cost to the system, detracting overall energy capture. In this project, a passive method of reducing flow separation with flexible airfoils is investigated. The flexible airfoils have shown to increase airfoil performance as compared to a rigid design. While the flexible airfoils can increase airfoil performance, it cannot withstand the same wind speeds as its rigid counterpart. The performance improvement is speculated to be a result of boundary layer reattachment post the point of stall, reducing the flow separation and increasing lift when compared to the rigid design.

\textsuperscript{1} National Science Foundation

NP05.00094 ABSTRACT WITHDRAWN –

NP05.00095 ABSTRACT WITHDRAWN –

NP05.00096 DFD POSTERS –

NP05.00097 Numerical Study of the Hydroacoustic Characteristics of a Marine Propeller, VIJAYAKUMAR RAJAGOPALAN, VIJIT MISRA, DANIO JOE, Indian Institute of Technology Madras — Propulsion of surface and under water ships with low noise characteristics depends on a number of factors in terms of their safety and operational performances, and it is crucial to predict and control their underwater noise characteristics. In this respect, the main scope of this study is to calculate numerically the propeller noise, which is one of the main sources of underwater noise. Therefore, propeller noise is studied numerically for non-cavitating conditions. Flow around the propeller is solved with a commercial CFD software, while hydro-acoustic analysis is performed using a model based on Flowcs Williams-Hawking equation. Flow around a propeller is solved using a RANS solver with the SST k-\omega turbulence model. Then, transient solution is performed with second order implicit pressure-based solver. Velocity and pressure coupled via SIMPLE algorithm Numerical Methods and Flow Solver. Time dependent pressure data is used as the input for the FWH equation to predict far-field acoustics.

NP05.00098 Analysis of Unsteady Wall Jet Created by a Coaxial-Rotor in Ground Effect, VRISHANK RAGHAV, LOKESH SILWAL, Department of Aerospace Engineering, Auburn University — The study of unsteady characteristics of the outwash of a coaxial-rotor configuration droning at a fixed axial separation distance operating at a tip Reynolds number around 150,000. The effect of axial separation distance on the instantaneous interactions between the tip vortex structures in the outwash region and its influence on the dynamics generated in the outwash is investigated. The variation of the outwash characteristics with varying ground heights for the drone is also discussed.

NP05.00099 Circular towing: a restricted case, JAMES HANNA, University of Nevada, Reno — Dynamic equilibria of strings in fluids are relevant to a variety of situations including fiber sedimentation and cable towing. Here I consider a highly restricted special case of steadily rotating planar configurations experiencing purely normal linear drag forces, for which analytical results may be obtained. The resulting shapes approach Fermat spirals at sufficiently large radius.
NP05.00100 Analysis of the self-starting capability of a new hybrid vertical axis wind turbine with a fluid-structure interaction approach¹, MEILIN YU, KAN LIU, WEIDONG ZHU, University of Maryland, Baltimore County — Vertical axis wind turbines (VAWTs) provide promising solutions for distributed wind energy harvesting in both urban and rural areas. However, it is challenging to guarantee satisfactory self-starting capability and high power efficiency simultaneously in a VAWT design. We have recently designed a new hybrid Darrieus-Modified-Savonius (HDMs) VAWT to address this challenge. The aerodynamics of the new hybrid design is analyzed using a fluid-structure interaction approach based on high fidelity computational fluid dynamics. We find that compared to the Darrieus VAWT, the HDMs design has better self-starting capability due to the torque provided by the inner MS rotor at small tip speed ratios (TSRs); the HDMs design can maintain high power efficiency at large TSRS with an appropriately sized MS rotor. The key flow physics is that the HDMs design can keep accelerating at small TSRS due to the inner MS rotor, and can suppress dynamic stall on the Darrieus blades at large TSRS. The effects of the turbine configuration, inertia and loading on the self-starting capability and power efficiency are further studied.

¹The authors gratefully acknowledge the support of the Maryland Higher Education Commission through the Maryland Offshore Wind Energy Research Challenge Grant Program.

NP05.00101 Dynamics of overshooting convection in a rotating spherical shell, LYDIA KORRE, Laboratory for Atmospheric and Space Physics, University of Colorado Boulder, NICHOLAS A. FEATHERSTONE, Department of Applied Mathematics, University of Colorado Boulder — Overshooting convection is a physical process by which turbulent convective motions generated in a convectively unstable region can propagate into a stably stratified zone that lies either on top or on bottom of the convective one. This process can lead to mixing of chemical species, thermal mixing, as well as contribute to the transport of magnetic fields and angular momentum. Thus, convective overshooting has direct and significant implications in stellar dynamics. Motivated by the Sun, we investigate these dynamics via numerical simulations in a spherical shell containing a convective zone with an underlying stable region. We present results of our runs which span a range of parameters and illustrate the dependence of convective overshooting on the intensity of the turbulence and degree of stratification of the convective region, the relative stability of the stable zone, the transition width between the two regions, as well as the rotation rate. These results can be particularly useful for gaining a better understanding of convective overshooting processes in stars and for improving existing models prescribed in 1D stellar evolution calculations.

NP05.00102 Modelling nutrient delivery to cells grown in a multiscale perfusable system, MOHIT DALWADI, University of Oxford — Growing cells in vitro expedites the process of testing viable drugs and reduces the need for animal testing. However, current methods to grow 3D structures eventually result in the formation of a necrotic core due to lack of nutrient access. One way to circumvent this is to use 3D bioprinting and photopatterning techniques to engineer multiscale perfusable systems that enhance nutrient delivery to cells. While these techniques offer a high degree of control over the configuration of the perfusable channels, it is not clear how the channels or cells should be distributed in order to maximize nutrient transport and avoid necrotic zones. To tackle these problems, it is imperative to have knowledge of the fluid flow within the perfusable hydrogel system; advection is the dominant nutrient transport mechanism. Thus, understanding and being able to control the flow within the biotube is paramount. In this talk, we use mathematical modelling to investigate how the nutrient transport to the growing cells is affected by experimentally controlled parameters, such as channel distribution, cell density, and the flow rate of nutrient fluid through the perfusable hydrogel structure.

NP05.00103 Transition to collective motion in two-dimensional microswimmer suspensions, VIKTOR SKULTETY, ALEXANDER MOROZOV, School of Physics & Astronomy, University of Edinburgh — A collection of microswimmers immersed in an incompressible fluid is characterised by strong orientational interactions due to the long-range nature of the hydrodynamic fields generated by individual organisms. As a result, suspensions of ‘pusher’ swimmers exhibit a state often referred to as collective motion or ‘bacterial turbulence’, which is dominated by jets and vortices compromising many microswimmers. The onset of collective motion can be understood within a mean-field kinetic theory for dipolar swimmers. In 3D, the theory predicts that the instability sets in at the largest scale available to the suspension. Here, we present a mean-field kinetic theory for a suspension of dipolar swimmers confined to a 2D plane embedded in a 3D fluid. We analyse the stability of the homogeneous and isotropic state, and find two types of instability: one is the analogous of the orientational instability in 3D systems, while the other is associated with strong density variations absent in 3D. In contrast to 3D suspensions, both instabilities occur at the smallest possible scale, and we discuss their implications for the ensuing collective motion.

NP05.00104 ABSTRACT WITHDRAWN

NP05.00105 Coupled fluid-structure interaction and mass transport in aortic valves, MOHAMMADREZA SOLTANY SADRABADI, Northern Arizona University, IMAN BORAZJANI, Texas A&M University, AMIRHOSSEIN ARZANI, Northern Arizona University — Near-leaflet biotransport processes play an important in calcific aortic valve disease initiation and complications are studied. First, constant biochemical concentration is imposed at the aortic root. Next, constant biochemical flux is imposed model of an aortic valve is coupled to continuum advection-diffusion transport equations. Two classes of problems representing aortic valve bioprosthetic aortic valve thrombosis. The solution to these transport processes involves coupled blood flow, nonlinear structural mechanics, and convective mass transport problems. Herein, 2D simulations are carried out where a two-way coupled fluid-structure interaction (FSI) model of an aortic valve is coupled to continuum advection-diffusion transport equations. Two classes of problems representing aortic valve complications are studied. First, constant biochemical concentration is imposed at the aortic root. Next, constant biochemical mass flux is imposed at the moving leaflet. Subsequently, biochemical transport near the leaflet is studied. The results show a close connection between vortex structures and biochemical concentration patterns. Distinctions in concentration patterns on the aortic and ventricular side of the leaflet are shown and implications for calcification and thrombosis discussed.

NP05.00106 Force exerted by a Stokeslet on confining boundaries, ALEXANDER MOROZOV, VIKTOR SKULTETY, School of Physics & Astronomy, University of Edinburgh, UK — Solutions to the Stokes equation can be constructed by combining suitably placed Stokeslets and other singular solutions, that simultaneously satisfy the equation of motion and the boundary conditions. This approach has proven especially fruitful in describing the motion of small solid bodies and self-propelled particles. Recent debate on the pressure exerted by microswimmers on the walls of the enclosing container, together with the observations of the apparent viscosity of microswimmer suspensions being strongly affected by their presence, stresses the need to evaluate the forces exerted by microswimmers on solid boundaries. Here, we study two archetypal problems: a Stokeslet next to a single flat boundary, and a Stokeslet confined in-between to parallel walls. This allows us to calculate forces exerted on the walls by microswimmers, and we find that while in the case of a single wall microswimmers exerts no total force on the wall, the force becomes nonzero in the latter. We estimate the pressure exerted on the wall by the typical dilute bacterial suspension used in experiments.
NP05.00107 An implicit adaptive high-order flux reconstruction framework for scale-resolving simulation of unsteady flows over moving complex geometries1, MEILIN YU, LAI WANG, University of Maryland, Baltimore County — We present a recent development of an implicit adaptive high-order flux reconstruction framework for moving domain simulation at intermediate and high Reynolds numbers. In this framework, the high-order flux reconstruction method is used for spatial discretization; the explicit first stage, singly diagonal implicit Runge-Kutta (ESDIRK) method is used to perform time integration; a dual-time stepping approach is used to assist convergence while maintaining temporal accuracy; a matrix-free implementation of the restarted generalized minimal residual (GMRES) method is employed to solve the large linear system; and a flow-resolution-based p-adaptation algorithm is adopted to apportion the computational resources to critical flow regions. The arbitrary Lagrangian and Eulerian (ALE) approach is used to enable moving domain simulation with body-fitted unstructured meshes, and the radial basis function (RBF) interpolation is employed to handle mesh movement and deformation. Several challenging 2D and 3D unsteady flow cases in the moderate to high Reynolds number range are used to demonstrate the capability of the new numerical framework.

1The authors gratefully acknowledge the support of the Office of Naval Research through the award N00014-16-1-2735.

NP05.00108 Bio-inspired flows in unsteady environments. Part II: crosswind gusts. MEILIN YU, NAKEIN POUD, University of Maryland, Baltimore County, JOHN HRYNUK, U.S. Army Research Laboratory — Autonomous underwater vehicles (AUVs) and unmanned aerial vehicles (UAVs) usually need to carry out tasks in unstructured and dynamic flow environments. This poses a number of challenges that cannot easily be addressed by approaches developed for highly controlled environments, such as uniform flows frequently used in experiments and numerical simulation This work studies the impact of crosswind gusts on the performance of flapping wings/fins at moderate to high Reynolds numbers (i.e., in the range from 10e+4 to 10e+6). A high-order accurate flux reconstruction flow solver with moving/deforming body-fitted unstructured meshes is used to perform the numerical simulation. We find that dynamic stall in a crosswind gust is very different from the stalled flow at a large geometric angle of attack (AoA) due to the different transient dynamics of the leading and trailing edge vortices. Reynolds numbers can significantly affect the vortex structures over the suction surface of the foil The effects of relative position between the gust and foil and gust strength are also discussed in this study.

NP05.00109 Bio-inspired flows in unsteady environments. Part III: mean flow shear. MEILIN YU, University of Maryland, Baltimore County, Z.J. WANG, SAEED FAROKHI, University of Kansas — Autonomous underwater vehicles (AUVs) and unmanned aerial vehicles (UAVs) usually need to carry out tasks in unstructured and dynamic flow environments. This poses a number of challenges that cannot easily be addressed by approaches developed for highly controlled environments, such as uniform flows frequently used in experiments and numerical simulation This work studies the impact of mean flow shear on the performance of flapping wings/fins. A hyperbolic tangent mean flow shear profile is superposed onto the uniform freestream to generate a shear flow A high-order accurate spectral difference flow solver with moving/deforming body-fitted unstructured meshes is used to perform the numerical simulation, and dynamic mode decomposition is applied to analyze coherent flow structures. We find that flapping motion can significantly promote unsteady lift generation in mean flow shear; the stronger the shear is, the larger the lift is. At the same time, the lift coefficient is much larger than that predicted by the Kutta-Joukowski theory under the same flow conditions. Thrust generation is almost not affected by the mean flow shear.

NP05.00110 Non-Newtonian effects on the slip and mobility of a self-propelling active particle, AKASH CHOUDHARY, PUSHPAVANAM S, IIT Madras — Self-propelling Janus particles generate concentration gradients along their surface by exploiting the asymmetry in surface activity. This gives rise to a ‘slip’ at the particle surface, which propels the particle without the requirement of external concentration gradients. In this work, the influence of viscoelasticity (second-order-fluid model) and shear-thinning/thickening (Carreau model) on the slip and mobility of an axisymmetric active particle. Using matched asymptotic expansions, we provide an analytical expression for the modification of slip. Using reciprocal theorem, we demonstrate the influence of fluid rheology on particle mobility for low Peclet numbers. The current study also provides insights into the transport of complex fluids through phoretic pumps.

NP05.00111 Computational simulation of guidewire motion in a blood vessel1, WANHO LEE, National Institute for Mathematical Sciences — The guidewire is made of a thin stainless-steel wire, inserted into the human body and moved through the blood vessel, and is an essential tool for the treatment and diagnosis of vascular diseases. In this study, Kirchhoff rod theory is applied to develop a guidewire model as an elastic rod, and to simulate moving within a given blood vessel. Particularly, the inherent characteristics (shape, strength, torque, and elasticity) of the guidewire are applied to the model, and the reaction of the guidewire to the axial movement and rotation of the operation portion is simulated. The blood vessel is presented with single branch, and the movement of guidewire along the shape of the vessel is examined. It will be also discussed the tip shape of the guidewire that must be selected to navigate to the desired path. The development of guidewire simulations can provide a safe environment for practitioners to practice as often as necessary while avoiding bioethics issues. In addition, it is possible to find an optimal pathways and controls for moving the guidewire to the clinical target with minimal stress on the environments within the vessel.

1This work was supported by the National Research Foundation of Korea grant funded by the Korea government (2017R1E1A1A03070847).

NP05.00112 A Mechanism by Which Nose Bluntness Suppresses Second-mode Instability. JOSEPH KUEHL, ARMANI BATISTA, ARHAM AMIN KHAN, University of Delaware — A physical mechanism by which nose bluntness suppresses second-mode instability is proposed. Considered are 7 degree half-angle straight cones of various nose bluntness radii at tunnel conditions relevant to the AFOSR-Notre Dame Large Mach 6 Quiet Tunnel (Lakebrink et al. 2018). It is shown that second-mode suppression is achieved via entropy layer modulation of the basic state density gradient. A weakening of the density gradient disrupts the acoustic resonance necessary to sustain second-mode growth. These results are consistent with the thermoacoustic interpretation (Kuehl 2018) which posits that second-mode instability can be modeled as thermoacoustic resonance of acoustic energy trapped within an acoustic impedance well. Furthermore, the generalized inflection point criteria of Lees and Lin (1946) is applied to develop a criteria for the existence of second-mode instability based on the strength of the basic state density gradient. Lakebrink, M., T., K. G. Bowcutt, T. Winfree, C. C. Huffman and T. J. Julian, 2018. Journal of Spacecraft and Rockets. 55 (2): 315-321. Kuehl, J. 2018. AIAA Journal, 1-8, 10.2514/1.J057015. Lees, L., and Lin, C.-C., 1946. NACA TR 1115.
NP05.00113 Observations of Thermodynamic and Kinematic Properties During the Morning Transition in a Wind Turbine Array Boundary Layer Using an Instrumented Unmanned Aerial Vehicle . KEVIN ADKINS, Embry-Riddle Aeronautical University, ADRIAN SESCUC, Mississippi State University, CHRISTOPHER SWINFORD, NIKOLAUS RENTZKE, Embry-Riddle Aeronautical University — Observation, simulation and modeling have shown that wind farms alter downstream atmospheric properties as turbulent wakes generated by turbines enhance vertical mixing. With a large portion of wind farms hosted within an agricultural setting, changes to the wind turbine array boundary layer (WTABL) are important as they can potentially impact crop productivity, along with inflow to downstream turbines. The authors, and others, have demonstrated changes to thermodynamic properties within the WTABL during daylight observations made by small unmanned aerial systems (sUAS). The attainment of permission to fly at night and at higher altitudes, along with the enhancement of the sUAS instrumentation suite with fast-response three-dimensional sonic anemometers, enabled observations during predawn hours and through the morning transition. This work details observed changes to thermodynamic and kinematic properties during a series of overnight field campaigns undertaken during the summer of 2019 around a utility-scale wind turbine.

NP05.00114 Experimental study on resonance heat generation due to shock wave compression1 . SEONGHYEON SEO, Hanbat National University — The presentation addresses an experimental study of heat generation phenomenon associated with the conversion of flow kinematic energy to thermal energy through acoustic resonance. The phenomenon has been examined using a set of a sonic nozzle and a resonance tube. An underexpanded jet from the nozzle enters the tube and its shock waves compress the gas in the tube and reflect from the end. The repetition of the compression in a resonant manner results in an abrupt increase of gas temperature inside. The results indicate that various geometrical and flow parameters including a nozzle diameter, a tube inlet diameter, a distance between the nozzle and the tube, a nozzle stagnation pressure, a mass flow rate and a working fluid, affect heat generation characteristics. High-speed Schlieren imaging of supersonic flow in the vicinity of the nozzle exit and the tube inlet successfully identifies and retraces the space adjacent to the nozzle edge to allow the reflecting flow to expand and escape the tube for the occurrence of the resonance. The application of helium as working gas compared with nitrogen shows that the temperature increase becomes two times greater and it reaches beyond 1000 K in less than two seconds.

1Hanbat National University

NP05.00115 Stability of Semi-Extrapolated Finite Difference Schemes . NARSHINI GUNPUTH, MIKAYLA FELDBAUER, SHEILA WHITMAN, ANDREW BRANDON, Lycoming College — When numerically solving partial differential equations, finite difference methods are a popular choice. Several factors come into play when choosing a finite difference method, such as stability, accuracy, and computational cost. In response to the small stability regions of explicit methods and the computational cost of implicit methods, we've developed a novel discretization technique called semi-extrapolation. Semi-extrapolation generates explicit schemes from implicit schemes by applying extrapolation in an unconventional fashion. Extrapolation can severely curtail stability, however, we've found that semi-extrapolation can improve stability, as compared to analogous explicit methods. In our presentation, we'll introduce our semi-extrapolation technique and discretize the Advection-Diffusion Equation according to semi-extrapolated and mainstream methods. Then, we'll discuss the stability regions of the schemes and analyze how the stabilities of the semi-extrapolated schemes compare to the stabilities of analogous schemes.

NP05.00116 Accuracy and Computational Cost of Semi-Extrapolated Finite Difference Schemes . SHEILA WHITMAN, MIKAYLA FELDBAUER, NARSHINI GUNPUTH, ANDREW BRANDON, Lycoming College — When numerically solving partial differential equations, finite difference methods are a popular choice. Several factors come into play when choosing a finite difference method, such as stability, accuracy, and computational cost. In response to the small stability regions of explicit methods and the computational cost of implicit methods, we've developed a novel discretization technique called semi-extrapolation. Semi-extrapolation generates explicit schemes from implicit schemes by applying extrapolation in an unconventional fashion. Semi-extrapolation can improve stability, however, we've also found that semi-extrapolation can have unexpected and interesting effects on accuracy. In our presentation, we'll introduce our semi-extrapolation technique and discretize the Advection-Diffusion Equation and the Advection-Diffusion Equation according to semi-extrapolated and mainstream finite difference methods. Then, we'll examine the computational costs and accuracies of semi-extrapolated methods. Included in this examination will be a comparison against the costs and accuracies of mainstream methods and a discussion regarding how stability influences the accuracy of semi-extrapolated schemes.

NP05.00117 State-free Front Tracking for Compressible Multi-material Problems . DANAIL VASSILEV, JAMES PECOVER, NICHOLAS HAWKER, NATHAN JOINER, ARTURAS VENSKUS, NICHOLAS NIASSE, THOMAS EDWARDS, JON HERRING, DAVID CHAPMAN, MARTIN READ, NIKITA CHATURVEDI, ADAM FRASER, First Light Fusion Ltd. — High energy density physics (HEDP) is a rapidly growing field studying interaction of matter and energy under conditions of extreme temperature, pressure and density. Numerical models capturing hydrodynamic instabilities and shocks are of crucial importance for understanding HEDP and designing different experimental components. Interface tracking methods, and specifically the front-tracking method with a fixed Eulerian mesh and a moving Lagrangian interface, have been applied to these types of problems with a best-in-class success Glimm2002.

NP05.00118 A Modified Cut-cell Approach for Inclined Boundary Conditions on Computational Fluid Dynamics . SAYURI TANAKA, NAOKI SHIMADA, YOSIHIIDE MATOBA, Sumitomo Chemical Co., Ltd. — A simulation method of fluid dynamics based on cut-cell approach was modified to deal with distorted boundary conditions. Most conventional cut-cell methods focus on a flux control across computational grids. On the other hand, we combined interpolation of fluid velocity and re-defined wall shear stress by using normal distance and projected fluid velocity. This combination was implemented in the structured computational grid. In addition, Detached Eddy Simulation approach with non-slip wall and logarithmic wall function model were used for calculation on turbulent flow in this study. Two-dimensional laminar flow with an inclined channel and air flows around a cylinder in three-dimensional fields were calculated to demonstrate applicability of our approach. As a result, we were able to obtain fair flow fields without any numerical instability, and its calculation efficiency was much higher than that based on unstructured approach from the viewpoint of accessibility of computational memory.
NP05.00119 Hytrac: A Hydrodynamic Front-Tracking Code for the Study of High Energy Density Multi-Material Flows, NATHAN JOINER, DAVE CHAPMAN, NIKITA CHATURVEDI, THOMAS EDWARDS, ADAM FRASER, NICHOLAS HAWKER, ION HERRING, NICOLAS NIASSE, JAMES PECOVER, MARTIN READ, DAN VASILEV, ARTURAS VENSKUS, First Light Fusion Ltd. — Inertially Confined nuclear Fusion (ICF) is an established research field pursued in laboratories worldwide; most notably in the US National Ignition Facility. First Light Fusion (FLF) is exploring alternative ICF directions, with the prime focus being sustainable power generation. Hydrodynamics and mixing of materials are critical to the design of an ICF target, where high-energy densities bring additional complexities to CFD models. Interface-tracking is a challenging numerical problem in terms of accuracy and robustness. Hytrac, an AMR front-tracking code, was developed with the aim of overcoming these challenges, to enable reliable and robust iteration of complex target geometries and optimisation. It has been verified in detail, with standard compressible fluid tests and methods, and validated against in-house experimental capability. Hytrac is parallelised using HPX, for efficient load balancing. It includes state-free tracking methods, and the more established Glimm approach. It also supports fluid nodes of arbitrary order, includes thermal conduction, thermal nonequilibrium, advanced physics, exact and approximate Riemann solvers, and several algorithms for space reconstruction and explicit time solution.

NP05.00120 A parallel pore-scale multiphase flow tool using the lattice Boltzmann method, SAHAR BAKHSHIAN, SEYYED ABOLFAZL HOSSEINI, University of Texas at Austin — The main focus of our study is to mimic multiphase flow in realistic three-dimensional rock models that enables us to gain a better insight into the effect of pore-scale phenomena on real reservoir problems. We developed a fluid flow simulator using a D3Q19 multiphase multi-relation-time (MRT) lattice Boltzmann (LB) model. The present LB model is an extended Color-Gradient approach with improved numerical stability and can handle multiphase flow simulations with low capillary number and high viscosity ratio. To improve the computational efficiency of the LB simulations to a reasonable level for industrial applications, the model has been applied to a parallel scheme written in C++ using the Message Passing Interface (MPI). We herein introduce the capability of our tool for multiphase flow simulation in porous media and present its application to CO2 sequestration in geological formations. The model has been applied to the simulation of CO2 and brine in sandstone rocks, by employing three-dimensional micro-CT images of rock samples. Injection of supercritical CO2 into the brine-saturated rock sample is simulated and complex displacement patterns under various reservoir conditions are identified.

NP05.00121 A comparison between functional derivative-based global sensitivity analysis and mixed Karhunen-Loève active subspace analysis of flows through porous media with uncertain material properties, MEILIN YU, University of Maryland, Baltimore County, ALEN ALEXANDERIAN, HELEN CLEAVES, HAYLEY GUY, RALPH SMITH, North Carolina State University — Flows through porous media, such as aquifer and biological tissues, are usually affected by the uncertain material properties. Quantifying the flow uncertainty due to the material uncertainty, and identifying the flow sensitivity to material properties are important practices to understand the underlying flow physics. Recently we have developed a functional derivative-based global sensitivity analysis (GSA) method and a mixed Karhunen-Loève active subspace analysis method for surrogate modeling of models with high-dimensional inputs and functional outputs. We have applied these two approaches to analyze the pressure field from biotransport in porous tumors. We find that the functional derivative-based GSA method is effective in finding input parameters that are important to the parameterization of the pressure field from biotransport in porous tumors. We also propose a new scaling for tmax with respect to the impact velocity which is in agreement with the experimental observations. We then present a new way to collapse in a master curve the evolution of the micro to nanometer drop contact diameter during impact for different wettabilities and different impact velocities.

NP05.00122 Stable long-wavelength convection with Dirichlet thermal boundary conditions through a Batchelor-Nitsche instability, ALARIC ROHL, LAYACHI HADJI, The University of Alabama — It is a well known fact that the onset of Rayleigh-Bénard convection occurs via a long-wavelength instability when the horizontal wavelengths of the fluid motion are greater than a critical value. The present study considers the case of Rayleigh-Bénard convection in a cell of infinite extent in the horizontal direction and having vertical walls of arbitrary shape. We consider two cases, one with thermal boundary conditions and the other with no-slip conditions. The equation for the Rayleigh-Bénard convection is solved numerically using a finite difference method. We find that the critical wavelength for the onset of convection is independent of the shape of the vertical walls. We also find that the critical wavelength is sensitive to the Prandtl number and the Rayleigh number. We use lubrication theory to derive an equation for the critical wavelength, which agrees well with the numerical results.

NP05.00123 Insights into droplet impact shape dynamics, DURBAR ROY, SOPHIA M, Indian Institute of Science, RABIRRATA MUKHERJEE, Indian Institute of Technology, Kharagpur, SAPTARSHI BASU, Indian Institute of Science — The underlying mechanics related to the initial shape transition is being studied experimentally using high speed imaging techniques for a droplet impacting on various substrates. Two traditional (Glass and PDMS) and two bio-inspired surfaces were used. A very distinct change in the shape of the deforming droplet during the early phase of the droplet spreading was observed when the impact Weber number of the droplet was varied from 1 to 50. The number of the capillary waves (wavelength) on the droplet surface changes quite significantly. This shows the existence of a critical Weber number beyond which the shape of the droplet during the initial transient of the spreading phase changes. The wavelength of the capillary waves was found to be a function of the impacting Weber number. This work provides some basic insights related to this transitional behavior using basic dimensional analysis and scaling theories.

NP05.00124 Dynamics of droplet spreading on Non-Newtonian liquid films, GRIGORIOS-ATHANASIOS IOANNIDIS, GEORGE KARAPETSA, Aristotle University of Thessaloniki — We investigate the dynamics of a liquid drop as it spreads along the interface of a liquid film. We consider the case of a liquid layer which exhibits non-Newtonian characteristics and is described by the Ostwald de Waele constitutive law. In the limit of both a thin droplet and a thin subphase, we use lubrication theory to derive equations for the positions of the interfaces. We use a finite-element formulation to obtain numerical solutions of the evolution equations. The effects of the physical parameters and rheological characteristics on the interface shapes are studied.

NP05.00125 How wettability controls nanoprinting, JOEL DE CONINCK, University of Mons — Using large scale molecular dynamics, we study in detail the impact of nanometer droplets of low viscosity on substrates and the effect of the wettability between the liquid and the plate. We show the maximal contact diameter during the nanodroplet impact (Dmax) as well as the time required to reach it (tmax) agrees with experimental data at the macroscale showing similarities between droplet impacts at the nano and the macro scales. The comparison between the MD simulations and different models reveals that most of these models do not consider all the effects we observe at the nanoscale. Moreover, most of their predictions for the impact at the nanoscale do not correspond to the simulation results. We have developed a new model for Dmax which agrees not only with the simulation data but also the experimental observations and it also considers the effects of the liquid-solid wettability. We also propose a new scaling for tmax with respect to the impact velocity which is also in agreement with the experimental observations. We then present a new way to collapse in a master curve the evolution of the micro to nanometer drop contact diameter during impact for different wettabilities and different impact velocities.

1This project has received funding from the Hellenic Foundation for Research and Innovation (HFRI) and the General Secretariat for Research and Technology (GSRT), under grant agreement No 792
Controlling the pinning of a receding contact line in a flow coating process. JOEL DE CONINCK, University of Mons — Capillary flow coating is a simple and effective technique to print and assemble ordered nanoparticle-based structures over patterned surfaces. The technique makes use of a nanoparticle suspension confined between two plates. Solvent evaporation and sliding movement of the top plate induce an internal flow that leads to the accumulation of nanoparticles at the bottom receding contact line and to their deposition on the bottom plate. Nevertheless, a comprehensive understanding of the process remains elusive, and in this respect the dynamics of wetting at the receding contact line is known to play a critical role. With the help of large-scale molecular dynamics simulations, we investigate the dynamic contact angle at the receding contact line as well as contact-line pinning on substrate heterogeneities. We develop a model to predict the pinning time of a receding contact line as a function of the displacement speed of the top plate on both chemical and topographical heterogeneities. Confirmation of the dynamic nature of contact-line pinning and justification of the contact line settling time allow us to better describe the time evolution of the receding angle in presence of heterogeneities.

Hydrodynamics-Dominated Wetting Phenomena on Hybrid Superhydrophobic Surfaces. ARASH AZIMI, CHAE ROHRS, PING HE, Department of Mechanical Engineering, Lamar University, Beaumont, TX 77710 — On chemically heterogeneous rough surfaces, determining the true apparent contact angle is a challenging task, because the global minimum surface energy is hardly attainable. The equilibrium contact angle is associated with the global minimum surface energy of the droplet-air-substrate system. However, in practice, the most stable contact angle is the measurable contact angle of the system. We present a series of 3D simulations using various initial conditions to reach possible meta-stable states for a water droplet on four micro-patterned hybrid substrates. The surface energy and energy barriers are computed. The apparent contact angles compared with experiments, global contour of the droplets, and liquid-solid area fractions are presented. Our results reveal a hydrodynamics-dominated wetting behavior on these hybrid surfaces, and capture several meta-stable Cassie-Baxter states, which cause a large contact angle hysteresis. For a given micro-patterned hybrid substrate, a critical impact speed can be found, above which the impact method cannot overcome more energy barriers to reach a lower energy state. Furthermore, a smaller variation of the measured contact angles is observed on the substrate with a lower heterogeneity of topology and chemistry.

Spreading on viscoelastic solids: Are contact angles selected by Neumann’s law? STEFAN KARPITSCHKA, Max Planck Institute for Dynamics and Self-Organization (MPIDS), Göttingen, Germany, MATHIJS VAN GORCUM, Physics of Fluids Group, University of Twente, Enschede, The Netherlands, BRUNO ANDREOTTI, Laboratoire de Physique Statistique, Univ. Paris-Diderot, France, JACCO H. SNOELIJER, Physics of Fluids Group, University of Twente, Enschede, The Netherlands — The spreading of liquid drops on soft substrates is extremely slow, owing to strong viscoelastic dissipation inside the solid. A detailed understanding of the spreading dynamics has remained elusive, partly owing to the difficulty in quantifying the strong viscoelastic deformations below the contact line that determine the shape of wetting ridges. Here we present direct experimental visualizations of the dynamic wetting ridge, complemented with measurements of the liquid contact angle. It is observed that the wetting ridge exhibits a rotation that follows exactly the dynamic liquid contact angle, as was previously hypothesized [Karpitschka et al., Nature Commun. (2015)]. This experimentally proves that, despite the contact line motion, the wetting ridge is still governed by Neumann’s law. Furthermore, our experiments suggest that moving contact lines lead to a variable surface tension of the substrate. We therefore set up a new theory that incorporates the influence of surface strain, for the first time including the so-called Shuttleworth effect into the dynamical theory for soft wetting. It includes a detailed analysis of the boundary conditions at the contact line, complemented by a dissipation analysis, which shows, again, the validity of Neumann’s balance.

Droplet dynamics during condensation on tailoring nanostructured surfaces. HUA-YI HSU, CHUN-CHI LI, SHIH-YAO HUANG, Department of Mechanical Engineering, National Taipei University of Technology — Liquid condensation from the atmosphere on a solid surface is commonly found in nature. To enhance energy conversion, nanostructured surfaces are with high potential to utilize the water generating applications. In this work, the droplet condensation on the tailored surface has been investigated by a 2D phase field model which is favorable to study the interfacial dynamics under microscopic scale. To model the liquid extracted from the air, both liquid and vapor are initially coexisted in random distribution. Spinodal decomposition process is then used here in which the fluid starts from an unstable thermodynamic state, and the homogeneous phase separates into coexisting phases spontaneously. Initially the small liquid droplet generated along the nanostructured surface and gradually merged into a coalescence droplet. By analyzing the droplet formation, the phase change dynamics can be studied and its relation with the spatially distributed structures on surface. The overall performance enhancement created by surface nanostructured was examined in comparison to a flat surface. Our understanding of this work provides more insights into the nanostructured surface topography on mass and heat transfer to improve the energy efficiency.

Three-dimensional front-tracking model for evaporation of drops SAUL PIEDRA, CONACYT-CIDESI Centro Nacional de Tecnologías Aeronáuticas, ALFONSO CASTREJON-PITA, University of Oxford, EDUARDO RAMOS, UNAM, GRETAR TRYGGVASON, Johns Hopkins University — We present the development of a full three-dimensional model to simulate the evaporation of falling drops. The evaporation model is based on the simultaneous solution of the mass, momentum, energy and vapor mass fraction conservation equations for incompressible fluids, properly adapted to incorporate the possibility of mass transfer at the boundary between the phases. The mass reduction of the drop is influenced by local thermodynamic conditions which in turn are modified by the dynamics of the drop motion. The vapor mass fraction at the interface is computed through the Clausius-Claperyon relation. The set of equations is defined in the whole domain, including the interface, and are solved using the finite volume/front-tracking method. The solution of the resulting linear equations systems are solved using the CUSP library implemented in a GPU in order to reduce the computational time. The validation for the evaporative flux calculation was done by comparison with a one-dimensional analytical solution for the evaporation of a planar surface. The simulation results for a static drop showed very good agreement with the $d^2$ law. Simulations for a single and multiple falling drops are also presented.

SP and AACP acknowledge the support of the Royal Society via a Newton Mobility Award No. NI17019.
NP05.00131 Proper Orthogonal Decomposition Analysis of Turbulent Cryogenic Liquid Jet Injection Under Transcritical Conditions

DORRIN JARRAHBASHI, SALAR TAGHIZADEH, Texas A&M University, TEXAS A&M UNIVERSITY TEAM — Liquid-rocket and high-pressure diesel engines operate at pressures and temperatures that exceed the critical pressure and temperature of the liquid fuels during injection. The turbulent flow features and turbulent fluctuations are impacted by the transition from subcritical to supercritical conditions that in turn affect the turbulent mixing between the fuel and oxidizer in the combustor. LES and the proper orthogonal decomposition (POD) algorithm are employed to study the turbulent flow and dominant unstable flow modes at transcritical and supercritical conditions for cryogenic nitrogen jet injected into a warm nitrogen environment. The effects of the transition from transcritical to supercritical conditions on mixing layer behavior are investigated. Real-gas thermodynamic properties at supercritical conditions are considered via implementing the cubic Peng–Robinson equation of state and Chung’s method for obtaining fluid transport properties. The results show that the presence of large density gradients in the mixing layer at transcritical conditions smear out the turbulent coherent structures in the radial direction and turbulence shows anisotropic behavior near the mixing layer that retards the overall mixing process.

NP05.00132 Slip flow-enhanced streaming current in graphene oxide nanochannels

CHIH-CHANG CHANG, Department of Industrial Technology Education, National Kaohsiung Normal University, HUNG-WEI CHANG, RUEY-JEN YANG, Department of Engineering Science, National Cheng Kung University — In recent year, fast transport of water in carbon-based nanochannels due to the slip effect has attracted much attention. In this work, the pressure-driven streaming currents through sub-1nm nanochannels reconstructed by the layered material of graphene oxide (GO) was investigated experimentally and theoretically. The results show that the measured values of streaming current are 2–3 orders of higher than the predicted values calculated from electrokinetic model under no-slip assumption. It is inferred that the streaming current is greatly enhanced due to the presence of water slippage in sub-1nm partial wetting GO nanochannels. In addition, it is found that the slip length is strongly dependent on the KCl concentration, i.e., surface charge density. The estimated slip length is from 1 to 22nm. The lower surface charge density (KCl concentration) reveals the larger slip length. It is believed that our finding is beneficial to develop a higher efficiency of electro-kinetic power generator and electro-osmotic pump.

NP05.00133 Investigation of Corner Flows in Complex Supersonic Rectangular Jet Nozzles

SETH KELLY, TYLER VARTABEDIAN, EMMA GIST, DOMINIC DIDOMINIC, MARK GLAUSER, Syracuse University — The understanding of complex turbulent flows is of utmost importance when designing propulsive systems for next-generation-plus aircraft. Due to the highly unsteady nature of these flows, innovative techniques are required to extract the key physics insights that effect the overall performance of jet nozzles. The experiments in this study investigate the flow from a rectangular jet nozzle, specifically a Single Expansion Ramp Nozzle (SERN) over a proximal surface. The primary area of investigation is the flow in the corner regions and their influence on the downstream shear layers. The experiments conducted utilized a variety of measurement techniques including; Particle Image Velocimetry (PIV) to obtain high-resolution velocity measurements, high frequency response pressure transducers to measure the unsteady deck surface pressure, and acoustic measurements to detect the acoustic field acoustic measurements. These measurements help determine the effects of the jet corner regions on both the wall (deck plate side) and free (SERN side) shear layers. Additionally, this study aims to determine the influence of the corner flows on the downstream separation region as well as their role in near wall turbulence dynamics and unsteady deck loading.

NP05.00134 Development of multiple-color fluorescence image velocimetry

YUSUKE OTSU, JUN SAKAKIBARA, Meiji University, VIVEK MUNGUNDHAN, SIGURDUR THORODDEN, King Abdullah University of Science and Technology — Scalar image velocimetry (SIV) is the technique to extract velocity vectors from scalar field measurements. The usual SIV involves minimizing a cost function, that penalizes the deviation from the one scalar transport equation. This method can lead to multiple solutions and additional condition must be applied to select the best one if the full scalar gradient is zero over the volume. In addition, this technique is not applied for images with large displacement between two instances. Here we propose to minimize these problem with the reconstruction of the velocity field by using two different dyes and image deformation. Conceptually, we argue that having a double set of convergence criteria will result in a much more accurate velocity field. This improved SIV scheme is applied to the coaxial round free jet in liquid phase. The spatial velocity fields thus obtained demonstrated the good agreement of the velocity field solution with the continuity condition. The PDF of velocity fields also represent Gaussian distribution.

NP05.00135 Visualization of flow structure and its changes on flap with vortex generators

YOSHIYUKI ICHIKAWA, SHUNSUKE KOIKE, YASUSHI ITO, MITSUHIRO MURAYAMA, KAZUYUKI NAKAKITA, KAZUOMI YAMAMOTO, KAZUHIRO KUSUNOSE, Japan Aerospace Exploration Agency (JAXA) — For the aircraft design, the high-lift systems have an important role because of satisfying the demand for improving the lift for the take-off and landing configurations and increasing the payload. Vane-type vortex generators (VGs) are sometimes installed on the flap as a passive flow control device to improve the high-lift performances. However, the effect of VGs on the flow physics of the flap is still in discussion. In this study, the performance of VGs installed on the flap of a half-span model with a three-element high-lift wing was investigated by low-speed wind tunnel tests. In the tests, VGs were installed where chordwise velocity became nearly maximum on the flap and flow visualization was conducted with VGs in different size. The test results revealed that the interaction between longitudinal vortices generated by the VGs and cross flow on the flap influenced flow separation patterns behind the VGs, which depended on the size and installation spacing of the VGs. We also evaluated the lift coefficient of the wind tunnel model to investigate the relationship between the flow structures and aerodynamic performance of the flap with VGs.

NP05.00136 Control of flow and sound around leading-edge slat of 30P30N airfoil using plasma actuator

YUSUKE ONISHI, JUN SAKAKIBARA, Meiji university — We studied active control of flow and noise around a multi-element airfoil (30P30N) using time-resolved particle image velocimetry (TR-PIV), microphone and dielectric barrier discharge plasma actuator (DBD-PA). The angle of attack of the wing model fixed in the hard wall test section was 6° to 10°, and the Reynolds number based on the stowed chord length was 1.1 x 10^5 to 1.5 x 10^5. As a result of acoustic measurement, characteristic peaks were confirmed in the range of Strouhal number St = 1.0 to 4.0, which is based on the slat chord length. Periodic disturbance (St = 0.3 – 3.9) was applied to a shear layer formed from the slat cup using a DBD-PA placed at the lower of the slat. The noise peak frequencies were modified in synchronization with the disturbance frequencies of St = 1.0 ~ 2.8. These Strouhal number is close to the vortex shedding frequency in the case where the DBD-PA was deactivated. It was confirmed by TR-PIV and dynamic mode decomposition that the flow structures at the characteristic peak frequencies modified by the disturbance frequencies. The results show that the noise and the flow field around the leading edge slat could be controlled by the periodic disturbance.
NP05.00137 Reduction of drag acting on the Ahmed body using plasma actuators

FUKA MATSUMURA, RYOSUKE ODA, JUN SAKAKIBARA, Meiji University — Ahmed body is a scientific model of automobile, and it is known that the rear end of this model has an important role in aerodynamic characteristics such as drag and the wake structure. In this study, we used a dielectric barrier discharge plasma actuator array (DBD-PA) at the rear end of the Ahmed body with a slant angle of $\phi = 25^\circ$ and $35^\circ$ to control the drag acting on the model. The experiment was conducted in the wind tunnel at $Re = 3.95 \times 10^4$. The DBD-PA were uniformly installed in the spanwise direction at the edge between the slant surface at the rear end and the roof, and burst control was applied by changing the excitation frequency (burst frequency $f^+$) and the ratio (burst ratio, $BR$). As a result, at $f^+ = 100$ Hz, the drag coefficient tends to decrease as $BR$ increases. However, when $f^+ = 100$ Hz, where the burst frequency was large, drag was most reduced at $BR = 30\%$. The drag coefficient hardly decreased at $\phi = 35^\circ$. We hope to present a results of the one-dimensional array of the DBD-PA, which introduce disturbance distributed in the spanwise direction with the temporal phase difference allowing to create lambda type vortices into the shear layer.

NP05.00138 Transient Evolution of Flow Profiles in Shear Banding Wormlike Micellar Fluids

HADI MOHAMMADIGOUSHKI, PETER RASSOLOV, Florida State University, FLORIDA STATE UNIVERSITY TEAM — We report experiments on spatiotemporal evolution of flow field in model shear-banding viscoelastic micellar solutions in a Taylor-Couette cell. Our goal is to systematically study the effects of fluid elasticity on transient evolution of flow fields. By varying surfactant concentration, salt concentration, and temperature, we vary the fluid elasticity in the range $4.21 \times 10^{-4}$ to $8.57 \times 10^{-4}$ while keeping entanglement density fixed at $5.1 \pm 0.7$, viscosity ratio fixed at $40.3 \pm 2.9$ Po$\.s$, and curvature fixed at $0.085$. Our experiments show as shear strain increases, shear stress shows an overshoot followed by a decay towards steady state. Simultaneously with shear stress decay, fluid moves in the opposite direction to that of the imposed motion in a subset of the gap (i.e., back flow). Consistently with theory, the back flow strengthens as elasticity number increases. However, at very high elasticity numbers, the transient backflow disappears, contrary to the same theoretical predictions. In addition to the back flow, multiple shear band structures are observed in the transient flow at high elasticity numbers. These transient multiple bands persist to steady state. We surmise that the lack of back flows at high elasticity numbers is linked to the formation of transient multiple bands.

NP05.00139 Ultra-fine roughness effect on a flat plate boundary layer transition

AKIO YAKENO, HIROKI TAMEIKE, SHIGERU OYABASHI, Tohoku University — To reduce the viscous drag around an airfoil, delaying turbulence transition is one of the effective ways. In this study, we analyzed the influence of very small wavy roughness on the two-dimensional boundary layer transition with Direct Numerical Simulation, by resolving each small roughness. First, we found that the transition considerably delayed by $BR = 30\%$. Different excitation ratio (burst ratio, $BR$) as $f^+ = 0.1$ and $f^+ = 400$ Hz, the drag coefficient tends to decrease as $BR$ increases. However, at $f^+ = 100$ Hz, the drag coefficient hardly decreased at $\phi = 35^\circ$. We hope to present a results of the one-dimensional array of the DBD-PA, which introduce disturbance distributed in the spanwise direction with the temporal phase difference allowing to create lambda type vortices into the shear layer.

NP05.00140 Instability analysis of Poiseuille flow between two parallel walls partially obstructed by porous surfaces

NAMRATA ACHARYA, Naperville Central High School, SAMAN HOOSHYAR, PARISA MIRBOD, University of Illinois at Chicago — Plane Poiseuille flow is widely encountered in studies of diverse fields such as filtration, biomechanics, and geological problems. This work explores the effect of porous geometrical parameters, namely depth ratio and porous resistivity on the stability of Poiseuille flow over various porous surfaces. The most unstable mode is determined by numerically solving the eigenvalue problem derived from coupling between Navier-Stokes and Brinkman equations. Comparison of critical Reynolds number versus porous resistivity graphs for different depth ratios shows that there is an instability mode shift from the porous to the fluid with increasing the depth ratio. Also, the most stable mode occurs at smaller porous resistivity as the depth ratio becomes larger. To validate the theoretical analysis, we performed a set of experiments in which a Water-Glycerin solution flows through a channel partially obstructed by porous medium. The porous medium is modeled as square arrays of cylinders and is installed on the bottom wall of the channel. We found a good agreement between the steady-state and perturbed velocity profiles obtained analytically and experimentally.

NP05.00141 Radial distribution function of Lennard-Jones fluids in shear flows from intermediate asymptotics

LUCA BANETTA, University of Cambridge, ALESSIO ZACCONI, University of Milan, STATISTICAL PHYSICS GROUP TEAM — The microstructure of a suspension of particles is ruled by the probability of finding a reference particle in a position with respect to a target one, the pair correlation function. Its description under shear flow has been a challenge for theoretical methods due to the singularly-perturbed boundary-layer nature of the problem. Previous approaches have been limited to hard-spheres (HS) and suffer from various limitations in their applicability. Here, we present an analytic scheme based on intermediate asymptotics which solves the Smoluchowski equation with shear in spherical coordinates including both intermolecular and hydrodynamic interactions with the intent of describing the pair correlation function for realistically interacting particles in shear flows. First, the method has been validated through a comparison with the rdf of a HS fluid under strongly sheared conditions. Finally, we have been capable of studying the microstructure of a complex shear-thinning fluid such as the Lennard-Jones at varying values of the attraction strength: a new depletion effect is predicted in the microstructure of the LJ fluid under shear, a feature to our knowledge never discovered before.

NP05.00142 Development of lubricant impregnated organogel surface for sustainable high drag reduction

JAEHYEON LEE, GUN YOUNG YOON, SANG JOON LEE, Pohang Institute of Science and Technology — Lubricant-infused surfaces (LIS) where micro/nanostructured surfaces are infused with lubricating liquids have attracted much attention due to their slippery properties for drag reduction. However, most state-of-the-art LIS technologies require complex fabrication processes and suffer from depletion problem of the infused lubricant, which limit their scalability and sustainability of high performance. Thus, a new strategy is proposed to overcome these problems by utilizing lubricant-impregnated organogel surfaces (LIOS) for efficient and sustainable drag reduction. In this LIOS, a lubricating liquid is dispersed in a solid 3D cross-linked network assimilated through physical or chemical interactions. Owing to its distinctive liquid absorption and retention capacity, the proposed LIOS might work out the lubricant depletion problem. The LIOS exhibited an extremely low sliding contact angle of 1.2 $\pm$ 0.2, indicating a highly slippery surface. Due to the high slipperiness feature, the organogel had a large slip length of 209.4 $\mu$m, which gives rise to high drag reduction. The present results demonstrate a new strategy for LIS system of low cost and sustainable high drag reduction.

1 National Research Foundation of Korea [NRF-2017R1A2B3005415]
NP05.00143 Enhancement for long term integration results of one-way nested regional ocean model by employing boundary small eddy additions technique

JIN HWAN HWANG, PHAM VAN SY, Seoul National University — Generating small-scales structures during dynamical downscaling using the nested regional oceanographic models have been studied by the numerous studies. Those studies generated fine-scales feature in the nested regional circulation model with the large scale information at the lateral boundary incorporating with the local forcing. Recent studies however, found that the small-scales motions are under-developed seriously by downscaling work and such errors in the small scales can be transferred to the larger scales by inverse cascading and this could lead degrading the whole scale motions. This mainly limits the jump ratio of resolutions to small requesting the higher cost for downscaling. From that point of view, this work proposed an efficient technique, named boundary small eddy additions (BSEA) to enhance the quality of simulation and allow the higher jump ratio of the down scaling. The BSEA adds artificially the small-scale motions to the boundaries of a nested model based the spectral information of large-scale and can greatly improve the quality of the reproduction, even at much higher spatial resolution difference between the driving and the nested models.

1Korea Polar Research Institute

NP05.00144 1.5-layer flow along a slope and western boundary

JOSEPH KUEHL, CHARLES MCMHAON, University of Delaware, VITALII SHEREMET, NOAA — Analytic and semi-analytic solutions are derived for two important classes of layered geophysical flows: a topographically controlled flow along a slope and a western boundary current. Specifically, a similarity solution approach is used to solve the 1.5-layer shallow water equations. Case A: An analytic solution taking the form of an inverse tangent function is found to describe the flow of a bottom intensified (lower-layer), weak current moving along a broad shelf/slope. Case B: An ordinary differential equation is derived to describe the flow of a 1.5-layer (upper-layer) viscous western boundary current. This equation is solved numerically to study the effect of a deformable layer interface on the structure of the western boundary current and the results are compared with rotating tank laboratory experiments. Both cases are formulated as idealized, two-layer, rotating fluid basins with sloping bottom topography. Kuehl, J. J. 2014. Geophysical Research Letters, 41, Ibanez, R., J. Kuehl, K. Shrestha and W. Anderson 2018. Nonlinear Processes in Geophysics, 25, 201-205. Kuehl, J. J. and V. A. Sheremet 2014. Journal of Fluid Mechanics, 740 97-113. Zavala Sanson, L. and G. J. van Heijst 2002. Journal of Fluid Mechanics, 471, 239-255.

NP05.00145 Measuring Energy Flux using PIV Data

JIA, CHARLOTTE MABBS, BRUCE RODENBORN, Centre College — Determining the energy flux of an internal wave from the experimentally measured velocity field was made possible by the work of Lee et al. (Lee et al., Phys. Fluids, 26, 2014). This method is used in our work to measure the amount of energy dissipated when internal waves reflect from sloping boundaries by comparing the incoming energy flux to the outgoing energy flux through a surface near to the reflection region. We also use numerical simulations of the Navier-Stokes equations in the Boussinesq limit where the energy flux is known from the pressure and velocity fields. There is good agreement between our experimental and numerical simulation data, and we find that there are high rates of energy dissipation during reflection process at the critical angle when the boundary flows are strong. The results are consistent with Dettner et al. (Phys., Fluids, 25, 2013) who showed that strong boundary flows are excited by tidal motion over model topography, but the conversion of tidal energy into internal waves is weak.

1Louis Stokes Alliances for Minority Participation

NP05.00146 Cross-stream migration of a particle in a non-isothermal flow in a microchannel using inertial focusing

T. KRISHNAVENI, T. RENGANATHAN, S. PUS, Indian Institute of Technology, Madras — Cross-stream migration of particles is observed in an axial flow in the presence of finite inertia in inertial focusing. It is a passive separation method where particles migrate to equilibrium positions. These positions mainly depend on the two counteracting forces namely the wall lift force and the shear gradient force. The equilibrium positions can be altered by changing the velocity profile of the fluid. In this work, the migration characteristics of a particle in a parallel plate microchannel is studied numerically in a pressure driven flow under the influence of constant temperature gradient in the transverse direction. The viscosity variation in the lateral direction is considered since the fluid viscosity is dependent on the temperature. An immersed boundary method is used to study the particle migration. The particle equilibrium positions depend on the applied temperature gradient. The equilibrium positions are not symmetric since the velocity profile is not symmetric. The effect of temperature gradient, Reynolds number and particle size on the particle migration are analyzed.

NP05.00147 Van Hove Singularities due to hydrodynamic interactions among the spheres in two dimensional flow system

HYUK KYU PAK, IMRAN SAEED, TSVI TLUSTY, IBS Center for Soft and Living Matter/UNIST — Dispersion relation for the phonon-like collective vibration modes due to the hydrodynamic interactions among the spherical particles with periodic separation a in quasi two dimensional flow shows peaks at the wavelength of 4.759a. In analogy to Van Hove singularities in solids, the density of states for hydrodynamically interacting systems becomes infinite at this wavelength with vanishing group velocities. Existence of these singularities is verified by computer simulation of hydrodynamic phonons in two cases: for periodic boundary conditions and for the system with broken symmetries. However, in unbounded systems the collective vibration decays before reaching the singularity.

1This work was supported by Grant No. IBS-R020-D1 by the Korean government.

NP05.00148 The rising velocity of a slowly pulsating bubble in a shear-thinning fluid

MARCO DE CORATO, Institute for Bioengineering of Catalonia (IBEC), The Barcelona Institute of Science and Technology (BIST), YANNIS DIMAKOPOULOS, JOHN TSAMOPOULOS, Fluid Mechanics and Rheology Laboratory, Department of Chemical Engineering, University of Patras — We study the rising motion of small bubbles that undergo contraction, expansion or oscillation in a shear-thinning fluid. We model the non-Newtonian response of the fluid using the Carreau-Yasuda constitutive equation, under the assumptions that the inertia of the fluid and of the bubble are negligible, and that the bubble remains spherical. These assumptions imply that the rising velocity of the bubble is instantaneously proportional to the buoyancy force, with the proportionality constant given by the inverse of the friction coefficient. We evaluate the friction coefficient as a function of the rheological parameters and of the instantaneous expansion/contraction rate of the bubble. Our results show that the radial motion of the bubble reduces the viscosity of the surrounding fluid, and markedly decreases the friction coefficient of the bubble. We find that the average rise velocity of a bubble undergoes radical pulsations is increased. We compare our predictions with the experiments performed by [S. Iwata, Y. Yamada, T. Takashima, and H. Mori. J. nonNewton. Fluid Mech., 151(1-3):30–37, 2008], who found that the rise velocity of bubbles that undergo radial pulsations is increased by orders of magnitude compared to the case of bubbles that do not pulsate.
NP05.00149 Strategic placement of an obstacle eliminates droplet break-up in the flow of a microfluidic concentrated emulsion. ALISON BICK, JIAN WEI KHOR, YA GAI, SINDY K. Y. TANG, Stanford University — Droplet microfluidics has enabled a wide range of high throughput applications, such as digital polymerase chain reaction (dPCR) and antibiotic screening. However, few studies have attempted to increase the throughput of the droplet interrogation process. Previously, strategic placement of a circular post near a narrow exit reduced conflict between interactions among living organisms or particles. Inspired by such work, we placed a circular post close to the constriction entrance of a tapered microchannel. The results of our experiment demonstrate that the effects of this placement on droplet break up are noteworthy. If the obstacle position and size is properly selected, the probability that the droplet will break decreases by up to 99%, thereby enabling a 3-fold increase in droplet interrogation rate. Droplet break-up depends on drop-drop interaction and drop deformation. Optimal obstacle placement immediately before? the constriction reduces drop deformation, which in turn reduces drop break-up. Strategic obstacle placement is therefore an attractive strategy for increasing droplet throughput.

NP05.00150 Parametric Study of the Wetting Transition of a Moving Meniscus.1 JIHOON KIM, Korea Institute of Ocean Science and Technology, JIN HWAN KO, Jeju National University, HEUNGCHAN KIM, HWAJUN LEE, Korea Institute of Ocean Science and Technology — In this study, we investigated the wetting transition of a moving meniscus in a grooved microchannel through a detailed parametric study based on measurement by an optical tool and micro-particle image velocimetry to avoid the transition in designing the microchannel. The parameters investigated were pitch, flow rate, and height of a microchannel. The contact angle, contact speed, and interfacial pressure difference were analyzed according to the parameters. We found that the pitch is most effective, the flow rate is moderately effective, and the height is least effective on that. The height even does not affect the contact angle because the solid-fluid interaction at the groove edge is stronger than the fluid-air interaction. As the critical correlation, the contact angle, which is dependent on the pitch and the flow rate, and the height affect the air pressure between the grooves, which governs the air penetration flux and mainly determines the wetting transition. Therefore, a powerful way to delay the wetting transition is to reduce the degree of air pressure variation, specifically with a low pitch and a tall height with a low flow rate. Eventually, understanding dominant input parameters in relation to the wetting transition will be very useful in the design stage of microfluidic applications.

NP05.00151 Natural gas supercritical properties: a multiscale molecular simulations study1 ALEXANDRO KIRCH, NAIYER RAZMARA, JULIO MENEGHINI, CAETANO MIRANDA, Universidade de São Paulo — A typical deficiency of continuum approach relies on the models describing averaged fluid properties. In particular, those models usually do not consider important molecular features occurring at the atomic level, which influences the macroscopic regime. A suitable strategy to describe these phenomena occurring over temporal and spatial scales is to combine the continuum mechanics with higher resolution methodologies within a multiscale scheme. In this context, molecular dynamics (MD) simulations provide reliable values to describe fluid basic properties. The obtained quantities could serve as input parameters to address multiscale issues observed in numerous scientific and industrial applications, including the Oil & Gas challenges. In present study, we take advantage of the predictive role handled by MD simulations to determine natural gas properties in the supercritical region, since there exist a lack of experimental data describing the mixture properties. In light of the multiscale approach, we discuss how the gas density, viscosity, and diffusion could feed lower resolution methodologies to address Oil & Gas interest systems, including nanoflows in porous media and membranes technology for gas separation.

1This research was a part of the project titled Infrastructure project for underwater construction robot test(2nd), funded by the Pohang City in Korea.

NP05.00152 Single Pixel Resolution Optical Flow for Low-rank Flow Fields1 TAKU NONOMURA, Tohoku University, Presto, JST, SHUNSUKE ONO, Tokyo Institute of Technology, Presto, JST — Single pixel resolution optical flow for quasi-steady fluid motion is proposed, while the conventional methods have been smoothed temporally or spatially. The proposed method does not use any spatial or temporal smoothing but utilizes the prior information that the fluid fields can be expressed by low-rank dataset. The new objective function with the restriction of low-rank approximation is formulated and solved by fast iterative shrinkage thresholding algorithm (FISTA) and randomized singular value decomposition (rSVD). The use of both FISTA and rSVD help us to speed up to solve the optimization problems. This algorithm gives us the single-pixel resolution flow fields form the pair images of flows, such as particle images. In the presentation, the details of objective functions and the results of numerical experiments will be reported.

1This research is partially supported by Presto, JST (JPMJPR1678).

NP05.00153 Characterization of drag coefficients of a sphere in non-Newtonian fluids by energy dissipation rate1 HAE JIN JO, WOOK RYOL HWANG, Gyeongsang National University, YOUNG JU KIM, Korea Institute of Geoscience and Mineral Resources — Accurate prediction of the drag coefficient of a single particle in non-Newtonian fluid is often required for analysis of fluid particle systems: for examples, prediction of behavior of rock fragments in drilling system, design and operation of slurry pipelines and solid-liquid separation processes. In this study, we introduce an effective viscosity in terms of energy dissipation rate to define the viscosity of the non-Newtonian fluid, and estimate the settling velocity and drag coefficient using effective viscosity. This method is established on the balance of the energy dissipation rate such that the external power is dissipated within the system as viscous dissipation in a laminar regime. The effective viscosity is a function of the effective shear rate, and the effective shear rate is determined by the apparent shear rate, which is the ratio of flow velocity to characteristic length, and the two flow numbers which depend geometrical characteristics of flow field only, almost independent of rheological property of a fluid. 1This work is supported by Korea Agency for Infrastructure Technology Advancement grant funded by Ministry of Land, Infrastructure and Transport (17IFIP-B133614-01, The Industrial Strategic Technology Development Program) and by the National Research Foundation of Korea (NRF-2019R1A2C1003974).

NP05.00154 ABSTRACT WITHDRAWN

NP05.00155 ABSTRACT WITHDRAWN
NP05.00156 Transition to turbulence in randomly packed porous media: energy and mixing characteristics. REZA M. ZIAZI, JAMES LIBURDY. Oregon State University — Transition to turbulence in randomly arranged porous media is observed in nature in applications such as cardiovascular, respiratory, and biological systems. The mechanisms driving the transition to turbulence through these flows are not very well identified. This work describes the parameters influencing on overall mixing during the transition process from the perspective of energy and dispersion characteristics by addressing the following questions: (a) what are the dominant mechanisms for energy growth emanating from swirl as opposed to turbulent kinetic energy budget, and (b) how does the inertial effects of vortical structures enhance the flow transport properties such as tortuosity and dispersion. Using time-resolved PIV, flow structures are investigated in the range of macro-scale Reynolds numbers from 100 to 1000 to show the pore- versus macro-scale effects on the energy of flow and swirl structures, turbulence production and dissipation, as well as dispersion, and their contribution in interpreting the overall flow mixing. We also show Lagrangian mixing characteristics based on Eulerian local pore velocity variances and relate these to macro-scale bed characteristics for uncovering the transitional processes in randomly distributed porous media flows.

NP05.00157 Gravity-Induced Ripening Undermines Capillary Trapping Stability. KE XU, Massachusetts Institute of Technology, YASHAR MEHMANI, Stanford University, LUORAN SHANG, Fudan University, QINGRONG XIONG, University of Manchester — Capillary (residual) trapping has long been considered as one of the safest CO2 sequestration mechanisms due of its hydrodynamic stability. Here we show, for the first time, that the long-term thermodynamic stability of capillary trapping could be compromised because of gravity, which has thus far been neglected in studies of bubble ripening. The interaction of gravity with molecular diffusion causes the vertical redistribution of trapped bubbles in a geologic porous medium, where bubbles at the top grow at the expense of bubbles at the bottom. The result is the formation of a gas cap at the top of the reservoir poised subsequent leakage risks. Accurate predictions of CO2 storage stability must therefore account for gravity-induced ripening. Here, we analyze the evolution of a population of trapped bubbles over time and develop simplified pore-scale and continuum models capturing its salient physics. The models reveal that the upward migration of CO2 may be hindered through the judicious selection of CO2 storage sites.

NP05.00158 ABSTRACT WITHDRAWN —

NP05.00159 Prediction of scattering properties for gas molecules on solid surfaces1. HIROKI KUSUNOSE, HIDEKI TAKEUCHI, National Institute of Technology, Kochi College — In high Knudsen number flows, thermal and flow properties of gas are strongly influenced by the characteristics of reflected gas molecules at solid surfaces. The investigation of scattering properties of gas molecules on solid surfaces is therefore important to analyze the flow fields for micro/nano scale flows or rarefied gas flows. However, the scattering properties of gas molecules depend on many factors such as atomic species, pressure and temperature in a flow field, and solid surface states. It is difficult to completely consider these factors for the investigation of the gas-surface interaction. The purpose of this study is to construct an effective model for predicting the scattering behavior of gas molecules on solid surfaces with adsorbate using machine learning approaches. The molecular velocity distribution functions of the reflected gas molecules were obtained by molecular dynamics simulations for the gas-surface interaction. The parameters of a Gaussian function model which expresses these velocity distribution functions for various adsorbed surfaces were predicted using machine learning. The velocity distribution functions based on the constructed model properly reproduce the results of molecular dynamics analysis.

1This work was supported by JSPS KAKENHI Grant Number JP18K03960.

NP05.00160 Fundamental Differences between Large-Eddy Simulation of Incompressible Turbulence vs. Premixed Turbulent Combustion. JAMES BRASSEUR, Univ of Colorado, YASH SHAH, Penn State Univ, PAULO PAES, Gamma Technologies, YUAN XIJAN, Penn State Univ — In contrast with RANS where the modeled terms are of leading order, the LES framework requires that the modeled subfilter-scale (SFS) contributions be of lower order than the leading-order terms. This will be the case if the resolved-scale (RS) contributions to the triadic sum of advective nonlinearities in spectral space dominate the SFS triads, requiring an effective grid that resolves well Reynolds stress motions. Turbulent combustion deviates from the LES framework is several key ways, primarily in the existence of the chemical source terms that lead to the release of thermal energy at scales generally unresolved. In this study we quantify the dominant SFS contributions to the key nonlinearities that underlie RS evolution in LES of premixed turbulent combustion to isolate fundamental deviations from the LES framework. With a new method to remove spurious spectral content in inhomogeneous directions, we apply a concurrent physical-Fourier space methodology to compressible DNS of flame-turbulence interactions to isolate the triadic structure of advective nonlinearities and the quadratic structure of chemical nonlinearities in the Fourier representation from which the dynamically dominant contributions are determined. We find that when the RS fluctuations in momentum and energy are well resolved, the relative SFS contributions are very different depending on the type of nonlinearity and the evolving RS vs. SFS content of individual species. Supported by AFOSR.

NP05.00161 Numerical Study of Detonations in Multiphase Flows. BENJAMIN J MUSICK, JACOB A MCFARLAND, Univ of Missouri - Columbia, PRASHANT TAREY, PRAVEEN K RAMAPRABHU, Univ of North Carolina Charlotte, DOUGLAS A SCHWER, Naval Research Laboratory Washington DC — The detonation phenomenon is of great interest in the engineering and scientific community. Much work has been done for gaseous detonations and their processes are relatively well understood. However, multidimensional, multiphase detonations develop characteristics that are more complicated to predict and understand. Many practical engineering applications aim to utilize liquid fuels due to their convenient nature, thus, a need arises for a greater understanding of liquid spray detonations. This poster focuses on the effects of varied initial conditions and physical models in two-dimensional liquid droplet JP-10 detonations. The effects of equivalence ratios, droplet size, droplet distribution, and particle breakup will be discussed. Different methods for particle tracking and generation will be discussed as well. Data was generated using the FLASH code developed by the Flash Center for Computational Science and modified for this work to include reactions and active particles using the particle-in-cell method. The multiphase results will be compared to data from gas phase simulations and other multiphase simulation codes.

NP05.00162 Space charge in pneumatically assisted electrospray. JULIA ZAKS, Agilent Technologies, TRYGVE RISTROPH — The importance of coulombic repulsion due to space charge on the trajectories of ions and droplets in pneumatically assisted electrospray depends on numerous physical properties of the spray. The influence of space charge has practical implications for systems that use electrospray, including ion sources for mass spectrometers. We develop a simplified theoretical model system representing an electrospray plume with high-pressure nebulizing gas. We solve Poissons equation and the continuity equation for this model system to find the radius of the plume, and use this result to quantify the effect of space charge on the trajectories of charged liquid droplets in the spray. These results are applied to ion sources for liquid chromatography-mass spectrometry by considering the effects of space charge on the trajectories of various mobile phases. The mechanisms driving space charge can be ignored in the case of modest electrospray currents, which correspond to low liquid flow rates. At higher currents and flow rates, the magnitude of the electric field from the space charge of the spray itself is comparable to that of electric fields from externally applied sources, suggesting that the role of space charge influences ion source behavior at high flow rates.
NP05.00163 Turbulent wall flows large-scale roughness heterogeneity: flow response to oblique alignment, WILLIAM ANDERSON, YIRAN ZHENG, UT Dallas — The physics of wall turbulence – ducts, boundary layers, and pipes – affects the aero-/hydro-dynamic signature of an array of flows. Such flows often occur under the “fully rough”, inertial-dominated limits for which viscous effects can be readily neglected and skin friction is driven by form drag (i.e., turbulent mixing). Rough surfaces composed of a complex height distribution are common in engineering and geophysical flows. Prognostic flow description is confounded by the presence of large-scale heterogeneity in surface geometry: that is, “patches” of differing roughness type, with spatial extent at least equal to the depth of the flow (i.e., duct half height, pipe radius, or boundary layer depth). When the prevailing transport direction is aligned orthogonal and parallel to such a heterogeneity, the flow responds with formation of an internal boundary layer or with counter-rotating rolls, respectively. These surface-driven secondary flows can completely disrupt outer layer dynamics, and thus have direct implications for wall modeled large-eddy simulation predicated upon outer-layer content. Moreover, realizations of precise orthogonal/parallel alignment are expected to be rare, and oblique alignment is likely the norm (for example, during vehicle maneuver). Results of wall turbulence response to such oblique arrangements are shown.

NP05.00164 Autonomous RANS/LES hybrid models with data-driven subclosures, GAVIN PORTWOOD, JUAN SAENZ, DANIEL LIVESCU, Los Alamos National Laboratory — We investigate the use of artificial neural networks (ANNS) to adapt classical Reynolds averaged Navier-Stokes (RANS) turbulence models for use in subgrid large-eddy simulation (LES) closures with the framework suggested by Perot & Gadebusch Phys. Fluids 19, 115105 (2007). In this study, we consider the application of a slightly-modified k − ε model to simulate stationary and decaying homogeneous isotropic turbulence (HIT) at a range of grid resolutions. These modifications dynamically account for (I) grid resolution relative to resolved motions and (II) backscatter from unresolved to resolved scales. In this framework, a modified turbulent viscosity accounts for the former, and the latter is determined empirically as a multiplicative factor in the modeled turbulent stress tensor. We leverage artificial neural networks to establish a universal form of this backscattering factor as a function of filter size and resolved flow statistics via a-priori analysis of direct numerical simulations. We perform simulations of stationary and decaying HIT at multiple grid resolution with the model and show via a-posteriori analysis that the use of an ANN to model complex physical phenomena, such as local upscale transfer, is both attractive and practical.

1This research was supported by the Advanced Simulation and Computing (ASC) program through the Physics and Engineering Models Mix & Burn (PEM-Mb) and the Advanced Technology Development and Mitigation Machine Learning (ATDM-ML) projects, and by LANL LDRD project #20190659DR. High-performance computing resources were provided via the ASC program at Los Alamos National Laboratory. Approved for unlimited release as LA-UR-19-27064.

NP05.00165 Magnus effect near flat ground, CHIN-CHOU CHU, HSIN-HUA LEE, Institute of Applied Mechanics, National Taiwan University, CHIEN C. CHANG, Institute of Applied Mechanics and Center for Advanced Research in Theoretical Sciences, National Taiwan University — This research is aimed to conduct experimental and numerical analysis of the Magnus effect when a circular cylinder is approaching a flat ground. The Reynolds number is fixed at 2000. Normalized parameters include the translation-rotation speed ratio α, declining velocity ratio β (translation-downward), and the gap ratio, denoted by SG =gap/D, where D=2cm is the diameter of the cylinder. The range of interest for α is from 0 to 0.2, and SG from 5 to 0.5. Three types of flow behaviors are identified according to the rotation of the cylinder: (I) non-rotating (α = 0), (ii) rotating counterclockwise (α > 0) and (iii) rotating clockwise (α < 0). In the first case (α = 0), the ground effect mitigates eddies behind the cylinder and leads to a higher lift and drag. In the second case (α > 0), as SG is decreasing, the lift and drag drops while the vortex shedding frequency increases. The vortex around the cylinder is alleviated by the ground effect, and the separation occurs at a lower portion behind the cylinder. In the last case (α < 0), as SG is decreasing, the drag increases while the vortex shedding frequency decreases. The vortex is strengthened by the ground effect, and the separation occurs at a higher location with the same reasoning. Further, stability analysis is applied to the three distinguished types of motion to examine their stability. In comparison, the phenomena of the flow patterns are consistent in both static and dynamic cases, yet the forces exerted on the cylinder are smaller in the dynamic cases.

1Supported by MOST, TAIWAN Grant No.s 105-2221-E-002-097-MY3 and 105-2221-E-002-105-MY3.

NP05.00166 Project micro-meddy: doubly-diffusive experiments with heated vortices, MICHAEL BURIN, ANDREW GONZALES, MARGARITA SANZ, CSU San Marcos, JOEL SOMMERIA, SAMUEL VIBOUD. Univ. Grenoble Alpes, CNRS, Grenoble INP, LEGI — We report on experiments that feature anticyclonic vortices embedded in linearly stratified, rotating tanks. When heated with respect to their surroundings there are two conspicuous instability features. First, prominent early, is that the circumferential edge appears serrated with cusp-like features from lateral intrusions, which are surmised to be due to thermal convection. Second, prominent later, a stepped layer develops above the vortex due to theromhaline diffusive convection. Observations are described from both the Coriolis platform (13m diameter tank, “1.5m diameter vortices”) and a prototype smaller vessel (0.3m diameter tank, “0.1m diameter vortices”). Our observations are considered with respect to previous laboratory work as well as to geophysical vortices that are thermally distinct from their enviorn, such as Atlantic Meddies.

NP05.00167 The trajectory of a leading-edge vortex following separation from an oscillating hydrofoil, YUNXING SU, QUENTIN GUILLAUMIN, KENNETH BREUER. Brown University — Oscillating hydrofoils operating at high angles of attack shed leading edge vortices (LEV) into the wake during their flapping cycle. Predicting the path that these vortices follow is critical when attempting to optimize the interactions between multiple foils operating in close proximity. Here, we report on Particle image velocimetry (PIV) measurements of the flow field generated by an oscillating hydrofoil at various pitch amplitude and reduced frequency. Using the Q-criterion (Haller 2005), the LEV location was identified and tracked both on the hydrofoil and in the wake. We find that a larger pitch amplitude generally resulted in an earlier LEV shedding from the foil together with the generation of a wider wake behind the foil; higher reduced frequency usually delayed the LEV shedding from the foil leading to a narrower wake. The effects of endplates were also explored. Here we find that that with endplates on the foil the LEV appeared more coherent and stayed closer to the foil. Once shed into the wake, the LEV shed with endplates generally travelled at the same cross-stream (Y) position, while the LEV shed without endplates continued to travel away from the centerline, resulting in a wider wake.
NP05.00168 Controlling the forces on tandem stationary cylinders using oscillatory rotational motion, RAVI CHAITANYA MYSA, Singapore University of Technology and Design, DOMINIC DENVER JOHN CHIDNDAR, Queen’s University Belfast, VINH-TAN NGUYEN INSTITUTE HIGH PERFORMANCE COMPUTING, ASTAR, PABLO VALDIVIA Y ALVARADO. Singapore University of Technology and Design — In a typical tandem circular cylinder set-up where the cylinders are held fixed, the forces acting on the downstream cylinder are primarily due to its own vortex shedding as well as the vortex interaction from the upstream cylinder. The upstream vortex upon interacting with the downstream cylinder displaces the boundary layer on it, which then leads to larger forces on the downstream cylinder compared to the upstream cylinder. The point of any stagnation point can however be controlled in a smart way by rotating the downstream cylinder in an appropriate manner. An oscillatory rotational motion with a frequency corresponding to the Strouhal frequency of the upstream cylinder is specified. However, the amplitude and phase of this rotational motion is actively monitored so that the forces on the downstream cylinder are controlled. A detailed analysis using the flow contours is conducted to explain the effect of oscillatory rotation motion on the forces of the downstream cylinder. Numerical simulations are performed at Reynolds number of 100 as well as at 0,000. This study will help in developing active feedback control for determining the forces acting on the downstream cylinder.

NP05.00169 Experimental Observation of Modulational Instability in Crossing Surface Gravity Wavetrains, JAMES N. STEER, The University of Edinburgh, MARK L. MCALLISTER, University of Oxford, ALISTAIR C. L. BORTHWICK, The University of Edinburgh, TON S. VAN DEN BREMER, University of Oxford — The coupled nonlinear Schrödinger equation (CNLSE) is a wave envelope evolution equation applicable to two crossing, narrow-banded wave systems. Modulational instability (MI), a feature of the nonlinear Schrödinger wave equation, is characterized (to first order) by an exponential growth of sideband components and the formation of distinct wave pulses, often containing extreme waves. Linear stability analysis of the CNLSE shows the effect of crossing angle, θ, on MI, and reveals instabilities between 0° < θ < 35°, 40° < θ < 143°, and 145° < θ < 180°. Herein, the modulational stability of crossing wavetrains seeded with symmetrical sidebands is determined experimentally from tests in a circular wave basin. Experiments were carried out at 12 crossing angles between 0° ≤ θ ≤ 88°, and strong unidirectional sideband growth was observed. This growth reduced significantly at angles beyond θ ≈ 20°, reaching complete stability at θ = 30°–40°. We find satisfactory agreement between numerical predictions (using a time-marching CNLSE solver) and experimental measurements for all crossing angles.

NP05.00170 Science for the People: The dual nature of Science, RODOLFO OSTILLA MONICO, Science for the People, SCIENCE FOR THE PEOPLE COLLABORATION — Science for the People is an organization dedicated to building a social movement around progressive and radical perspectives on science and society. We are STEM workers, educators, and activists who believe that science can be a positive force for humanity and the planet. Science is not an abstraction removed from society. Science is produced by our labor. But the conditions of this production and the use of science are not controlled by the general public. Science has been used historically against marginalized communities for example to justify racism. Today, we see anti-scientific movements such as anti-vaccines arising out of the mistrust between the scientific community and the people. We must ensure that science serves all people, including the most marginalized. Until this is addressed, the STEM field will lack diversity and any solutions to this are patchy. While knowledge is won with our labor and can be used to advance common goals, we recognize that technical knowledge alone never delivers justice. Because of this, our members are committed to educating scientists and university students about the role that science and technology play in shaping the world, and using our knowledge to work with specific grassroots campaigns.

NP05.00171 Creation and research at the arts & sciences chair in Paris, JEAN-MARC CHOMAZ, LADHYX, CNRS-Ecole Polytechnique — The “Arts & Sciences” Chair at the cole polytechnique, the cole nationale supérieure des Arts Dcoratifs-PSL and the Daniel and Nina Carasso Foundation was created in September 2017 with the ambition of bringing together the arts & sciences to imagine tomorrow, by promoting collaborative practices and a citizen approach. In a context of climate emergency and constant technological and digital developments, artists and scientists have in common that they can play a “scouting” role. The Arts & Sciences Chair offers a research and creation program to question our interdependence with the environment, human and non-human, whether it is to explore our relationship to plants and climate or human-machine interactions. Several examples of art installations involving rain, fog, oxygen, waves, instability and time will be presented.

NP05.00172 Data-driven design and predict heat exchanger performance: configuration complexity and difficulty of prediction, ZHIFENG ZHANG, Process Engineering and Combustion, Delaware Innovation Campus, American Air Liquide, Newark, DE 19702, YILUN CHEN, Computational and Data Science, Delaware Innovation Campus, American Air Liquide, Newark, DE 19702, JIEFU MA, Process Engineering and Combustion, Delaware Innovation Campus, American Air Liquide, Newark, DE 19702, COMPUTATIONAL AND DATA SCIENCE COLLABORATION, PROCESS ENGINEERING AND COMBUSTION COLLABORATION — Data-driven methods have shown potential capabilities in accelerating future designs, optimizations, and evaluations. However, due to the complexity of fluid flow and geometry configurations of industrial devices, it is challenging to train effective data-based models. In the present research, we explore the relationship between device configuration complexities and reliable prediction algorithms. Taking a 2D plate heat exchanger as an example, we first use computational fluid dynamics solver to generate fluid flow/heat transfer data and then we train machine learning models to predict device performance. By increasing the complexity of the geometry configurations, we repeat the process and study the relationship between geometry configurations and machine learning models. Through the study, we can conclude the suitable algorithm for the specific plate heat exchanger application.

NP05.00173 Effects of foil shape on fish-like swimming, COOPER KOVAR, MARGARET BYRON, AZAR ESLAM-PANAH, Penn State University — Aquatic animals have evolved a diversity of propulsive mechanisms to locomote effectively through water. Fishes produce hydrodynamic thrust by acceleration of water through movement of their body and tails, while simultaneously reducing the resistance to their motion through morphological design, phased kinematics, and behaviors. In this study, the effect of the shape of a fish-like caudal fin as well as changes in the frequency of the linear motion of the fin is investigated. Many numerical tests have been done due to optical limitations, and it’s time for experimental test to confirm the numerical data. Experiments were conducted on a square fin, triangle fin, and forked triangle fin in order to compare and observe the changes in efficiency and thrust produced. Furthermore, the amplitude and frequency of linear oscillations were varied to see the data at different speeds within the optimal Strouhal number range at which fish are more efficient. The results of this research can be used to confirm the data collected from numerical experiments and further be applied to future biomimicry.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P01 Waves and Free Surface Flows: Turbulence 2A - James H. Duncan, University of Maryland
We consider the long-term dynamics of nonlinear dispersive waves in a finite domain. This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

We consider the long-term dynamics of nonlinear dispersive waves in a finite domain. This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.

This configuration arises both naturally (e.g., surface waves in a finite tank) and computationally (e.g., when periodic boundary conditions are used to approximate an infinite system). While the finite domain effect on the formation of power-law wave spectra has been studied in the framework of discrete wave turbulence, the effect of the aspect ratio of the domain has remained unexplored. In this work, we study the long-term evolution of the two-dimensional MMT (Majda, Mclaughlin, Tabak) equation on both rational and irrational tori (mapped from two-dimensional planes with rational and irrational aspect ratios). It is shown that the dynamics (e.g., power-law spectra) of the two systems are remarkably different, especially at lower nonlinearity levels. The results are explained in the context of discrete nonlinear resonances, in particular by the survival of exact resonances on rational/irrational tori. We conclude by outlining the implications of these findings for other physical wave contexts.
6:21PM P01.00006 Surface Waves Enhance Particle Dispersion. MOHAMMAD FARAZMAND, THEMIS SAPSIS, MIT — How quickly does a point source of pollution spread in a fluid flow? Taylor’s single-particle dispersion theory predicts that, in homogeneous isotropic turbulence, the tracer variance $\langle |X(t)|^2 \rangle$ grows quadratically for short times $\langle |X(t)|^2 \rangle \sim t^2$ and linearly for large times $\langle |X(t)|^2 \rangle \sim t$. We show that these predictions break down for tracers dispersed on the free surface of a gravity wave. Using an exact nonlinear model to advect the tracer particles, we show that the nonlinear effects significantly enhance the dispersion. In particular, the tracer variance grows as $\langle |X(t)|^2 \rangle \sim t^4$ for times $t$ less than one wave period. In the asymptotic limit, as $t$ increases beyond one wave period, the variance grows quadratically with time, i.e., $\langle |X(t)|^2 \rangle \sim t^2$. We show that this super-diffusive behavior is a result of the long-term correlation of the Lagrangian velocities of fluid parcels on the free surface.

6:34PM P01.00007 Large-eddy simulation of oil droplet aerosol transport in wind over waves. DI YANG, MENG LI, University of Houston, YAJAT PANDYA, GIACOMO VALERIO IUNGO, University of Texas at Dallas — In this study, large-eddy simulations (LES) are performed to study the transport of aerosolized small oil droplets in wind over surface waves. The LES model consists of a wind turbulence solver and an Eulerian scalar transport solver. The wind LES solver uses a boundary fitted computational grid system that follows the instantaneous sea-surface waves to capture the turbulent flow structures and transport phenomena near the sea surface, and a hybrid pseudo-spectral and finite-difference method for spatial discretization. The aerosol transport is modeled using an Eulerian approach, in which an advection-diffusion equation for the oil droplet concentration field is solved using the finite-volume method with a combination of upwinding scheme and central difference scheme. Using this LES model, a set of simulations with various oil droplet sizes and sea-surface conditions are performed to investigate their effects on the vertical and lateral transports of oil aerosols in the marine atmospheric boundary layer.

1This research was made possible by a grant from The Gulf of Mexico Research Initiative.

6:47PM P01.00008 Froude number effects on free surface turbulence in open channel flow. YUJIA DI, University of Minnesota, Twin Cities; Shanghai Jiao Tong University, BINGQING DENG, ANQING XUAN, University of Minnesota, Twin Cities, YE LI, Shanghai Jiao Tong University, LIAN SHEN, University of Minnesota, Twin Cities — Direct numerical simulations of free surface turbulent flows in open channel at different Froude numbers have been conducted to study the surface deformation and the turbulence below. The deformation of the free surface is captured using a boundary-fitted grid and fully nonlinear free-surface boundary conditions. By studying the frequency-wavenumber spectrum of surface elevation, we find that the advection of the mean current alone contributes to the surface deformation at low Fr, while the contribution of the surface wave becomes more important as Fr increases. It is also found that in the bulk flow region, more energy of the streamwise velocity component is distributed at large length scales as Fr increases, indicating that the Fr number effect can also reach into the bulk region. The underlying mechanism of the modified bulk flows at high Fr numbers is further analyzed using the TKE budget spectrum.

7:00PM P01.00009 Scale effects for air entrainment in quasi-steady breaking waves. KELLI HENDRICKSON, DICK YUE, Massachusetts Institute of Technology — Quasi-steady breaking waves are prominent and highly observable features in civil, environmental, ocean and naval engineering applications with direct impact on turbulent dissipation and air-sea interaction. We use high-resolution 3D direct numerical simulation of quasi-steady breaking waves to study the air entrainment characteristics as a function of resolvable features within the wave. The numerical method utilizes conservative Volume of Fluid (cVOF) to capture the interface on a Cartesian grid. A submerged body generates the quasi-steady breaking wave. Our particular interest lies in developing parameterizations and models that relate the entrainment due to quasi-steady wave breaking to underlying flow characteristics. For different Froude numbers $Fr$, we observe two flow regimes: a periodic-wave-breaking and a metastable regime. For the periodic-wave-breaking regime we show that the bubble-size correlation of the Lagrangian velocities of fluid parcels on the free surface.

1Office of Naval Research

Monday, November 25, 2019 5:16PM - 7:13PM — Session P02 Multiphase Flows: Modeling and Theory III 2B - Andrea Prosperetti, University of Houston

5:16PM P02.00001 Interface Retaining Coarsening for Multiphase Flows. XIANYANG CHEN, JIACAI LU, GRETRAN TRYGGVASON, Johns Hopkins University Department of Mechanical Engineering — Multiphase flows are characterized by sharp moving phase boundaries, separating different fluids or phases. In many cases the dynamics of the interfaces determines the behavior of the flow. In a coarse, or reduced order model, either an averaged two-fluid model or a large-eddy-simulation like one, it is therefore critical to retain a sharp interface for the resolved scales. The point particle model for disperse flows is a widely used limiting case. Different strategies to retain sharp interfaces are possible. In the simplest case the indicator function identifying the different fluids is filtered and the sharp interface restored by nonlinear post-processing, consisting either of identifying the interface location from the filtered field, or letting the interface "flow to its new location. Another approach is to work directly with Lagrangian markers points identifying the interface and average their coordinates, or evolve the interface based on curvature or other measures. The different approaches are discussed and compared, the relationship with image processing in computer graphics pointed out, and implications for the flow field are studied. Modeling approaches for the unresolved scales are briefly reviewed.

5:29PM P02.00002 ABSTRACT WITHDRAWN —
both steady and transient flows. This study focuses on high fidelity simulations useful for the determination of both current study to develop a Drift-Flux model appropriate for mixing regions in pipe flows for a range of gas-liquid flow parameters that addresses, IVAN NEPOMNYASHCHIKH, JAMES LIBURDY, Oregon State University — The Drift-Flux model (DFM) is a powerful reduce order model that can be applied to gas-liquid flows. As applied to a one dimensional internal flow will be demonstrated and compared with experiments. Hydrodynamic, electrostatic and ultrasonic controlling strategies will also be modelled round-jet flow simulation according to experimental and theoretical statistics. Finally, Euler-Lagrange simulations incorporating these models will be demonstrated and compared with experiments. Hydrodynamic, electrostatic and ultrasonic controlling strategies will also be modelled to test their performance in changing the dispersion of spray droplets.  

1Supported by the Office of Naval Research (ONR) as part of the MURI Program, under grant number N00014-16-1-2617.

6:08PM P02.00005 Exploration of Gas-liquid Flow Mixing Region for the Purpose of Drift-flux Model Enhancement, IVAN NEPOMNYASHCHIKH, JAMES LIBURDY, Oregon State University — The Drift-Flux model (DFM) is a powerful reduce order model that can be applied to gas-liquid flows. As applied to a one dimensional internal flow the model requires specification of two parameters: \(C_0\) (distribution parameter) and \(v_s\) (weighted mean drift velocity). It is the goal of the current study to develop a Drift-Flux model appropriate for mixing regions in pipe flows for a range of gas-liquid flow parameters that addresses both steady and transient flows. This study focuses on high fidelity simulations useful for the determination of both \(C_0\) and \(v_s\). A wide range of both transient and steady flow conditions is used (inlet parameters and fluid properties). The result is a range of two phase flow conditions, or flow maps, within the mixing region. For each set of conditions results are used to identify values of \(C_0\) and \(v_s\). Functional relationships are then found to map these parameters to the flow map regimes. The extent and range of the functional relationships allows for a robust set of conditions that can be utilized in the Drift-Flux model. Using these results comparisons are made with existing functional forms of \(C_0\) and \(v_s\) to help identify the transient conditions of mixing.

6:21PM P02.00006 Nearest Particle Statistics and Particle-fluid-particle Stress of Multiphase Flows, DUAN ZHANG, Los Alamos National Laboratory — This presentation starts with showing an important relation between the ensemble average and the average based on the nearest particle. Using this relation one can study long-range particle-fluid-particle (PFP) interactions by defining an effective short-range correlation accounting for effects of other surrounding particles. Physical meanings of this correlation will be presented, and the mathematical derivations will be outlined. This nearest particle average is compared with the average conditional on a pair of particles. For short-range forces, the ensemble averages calculated from the both methods are the same. For long-range forces the nearest particle average has an advantage. Using the effective short-range correlation force for hydrodynamic interactions among particles, a PFP stress is defined. The physical meanings of this stress will be examined. Using dilute potential and Stokes flows as examples, the PFP stresses are calculated. For cases of finite particle volume fractions, a numerical method is proposed to calculate this stress using the results of particle forces. This study of nearest particle statistics leads to many interesting and unanswered questions of particle interactions in disperse multiphase flows.

1The financial support for this work is provided by the Advanced Simulation and Computing (ASC) program.

6:34PM P02.00007 A Physics/Data-Driven Comprehensive Multiphase Force Coupling Model That Systematically Accounts For Clustering & Shear Effects, GEORGES AKIKI, Notre Dame University-Louaize, DUAN ZHANG, Los Alamos National Lab, S BALACHANDAR, University of Florida — Traditional approach to modeling the phase interaction has been through drag and lift force models that depend on Re and volume fraction \(\theta\). However, it has been recognized that it is important to take into account local volume fraction gradients in particle distribution. Recently, it has been shown that the Particle-Fluid-Particle (PFP) stress can be rigorously defined, whose divergence accounts for the effect of inhomogeneous particle distribution. A series of fully-resolved DNS of flow around a uniform random distribution of particles are performed from which the PFP stress was evaluated. We then show how the DNS results can be accurately recovered with the pairwise interaction extended point-particle (PIEP) model. This allows the evaluation of the PFP stress for a wide range of Re and \(\theta\) at a negligible computational cost. We then tested the PFP force in inhomogeneous distributions of particles by comparing the model prediction against results obtained from DNS. We also performed DNS of shear flow through a uniform random distribution of particles to establish the shear-induced lift force model at finite \(\theta\). The significance of these new forces are then analyzed and compared to the mean streetwise force in uniform flows and to the mean streetwise and lateral forces in shear flows.

1We acknowledge DOE PSAAP-II (DE-NA0002378), ONR (N00014-16-1-2617), and Advanced Simulation and Computing program of Los Alamos.
6:47PM P02.00008 Resolved Simulations of Particulate Shear Flow – GEDI ZHOU, Johns Hopkins University, ANDREA PROSPERETTI, University of Houston — This work describes the results of a numerical study of a suspension of thousands of resolved spheres in shear flow carried out by means of the Physalis method. The particle/fluid density ratio is varied between 2 and 5, the volume fraction between 5% and 30% and the particle Reynolds number between 25 and 50. Gravity is disregarded. The results shown include the particle mean free path, the pair distribution function, the particle diffusivity, collision frequency, the mixture stress and others. The mean free path decreases as the particle density is increased. The significant particle lagging near the solid surface with increasing volume fraction, with a strong effect on the velocity distribution of the suspension and the wall shear stress. The collisional contribution to the stress becomes dominant as the particle density ratio and volume fraction increase.

7:00PM P02.00009 Comprehensive Analysis of Two Fluid Models for Turbulent Gas-Solid and Liquid-Solid Flow – OMAR RENZO PIMINCHUMO MARINOS, DONALD J. BERGSTROM, University of Saskatchewan — In the Eulerian Two Fluid Model (TFM) formulation, the transport equation for the disperse phase consisting of solid particles is obtained from the Kinetic Theory of Granular Flow (KTGF). The transport equation for the carrier fluid, either gas or liquid, closely resembles the single-phase Reynolds Averaged Navier-Stokes equation. Although successful predictions have been obtained for gas-solid flow, simulations of liquid-solid flow typically show less agreement with the experimental data. The present work compares a TFM developed for gas-solid flow with a recent TFM developed for liquid-solid flow: both describe the stresses of the solid phase using the KTGF. The main focus of the study is the difference in the predictions for the mean velocity, fluctuating velocity and volume fraction profiles, compared to experimental data for dilute liquid-solid flow in a vertical pipe. An outcome of the present research is a modification to the TFM which improves the predictions of the previous formulations. The modification considers changes to the model relations for the solid viscosity, granular temperature conductive coefficient and the source term intended to incorporate interstitial fluid effects.

Monday, November 25, 2019 5:16PM - 6:34PM – Session P03 Fluid Mechanics General 201 -

5:16PM P03.00001 Zermelo's problem: Optimal point-to-point navigation in 2D turbulent flows using Reinforcement Learning – MICHELE BUZZICOTTI, LUCA BIFERALE, University of Rome Tor Vergata and INFN, FABIO BONACCORSO, Center for Life Nano Science@La Sapienza, Istituto Italiano di Tecnologia, PATRICIO CLARK DI LEONI, Dept. of Mechanical Engineering, Johns Hopkins University, KRISTIAN GUSTAVSSON, Dept. of Physics, University of Gothenburg — To find the path that minimizes the time to navigate between two given points in a fluid flow is known as the Zermelo’s problem. Here, we investigate it by using a Reinforcement Learning (RL) approach for the case of a vessel which has a slip velocity with fixed intensity, \( \frac{V_s}{V_c} \), but variable direction and navigating in a 2D turbulent sea. We use an Actor-Critic RL algorithm, and compare the results with strategies obtained analytically from continuous Optimal Navigation (ON) protocols. We show that for our application, ON solutions are unstable for the typical duration of the navigation process, and are therefore not useful in practice. On the other hand, RL solutions are much more robust with respect to small changes in the initial conditions and to external noise, and are able to find optimal trajectories even when \( \frac{V_s}{V_c} \) is much smaller than the maximum flow velocity. Furthermore, we show how the RL approach is able to take advantage of the flow properties in order to reach the target, especially when the steering speed is small.

5:29PM P03.00002 Thermo-mechanics modeling and physics – DAVID KASSOY, Kassoy Innovative Science Solutions LLC, ADAM NORRIS, University of Colorado, Boulder — Thermal energy deposition into a finite volume of gas is the immediate source of thermodynamic and velocity disturbances. The thermo-mechanical response in a given geometry and system depends on a variety of physical parameters including the time scale and quantity of heat deposition into the volume, and on the time scale that characterizes acoustic disturbances in the volume. Thermo-mechanical modeling is based on the non-dimensional reactive Navier-Stokes equations. The relevant non-dimensional parameters include: \( \text{parda} \), the ratio of the characteristic time scale for heat release to the acoustic time scale of the volume, \( \text{parda} = \frac{\text{characteristic time scale for heat release}}{\text{characteristic time scale of the volume}} \). The traditional definitions of high activation parameter \( \text{parda} \) are based on the reactive Euler equations, are described for a wide range of non-dimensional parameters defined above. They are compared with asymptotic analytical results, obtained for a variety of limiting parameter values, that including non-intuitive nearly constant volume, nearly isobaric and fully compressible responses to energy deposition.

5:42PM P03.00003 Great operation – HAN YONGQUAN, 15611860790 — The even general formula is: \( 2n, \) where \( n \) is an integer greater than 1, and \( 2n \) can decompose the prime factor, that is, \( 2n=\text{prime numbers} \), when \( n \) is a prime number (specially pointed out: when \( n=2 \), \( 2n=2 \)), the proposition is proved. When \( m \) is a composite number, it is proved as follows: When \( m \) is an even number, \( m \) adds an odd number or subtracts an odd number to exhaust all odd numbers, and since more than 2 prime numbers must exist in the odd number, it is sure to find the sum of the two prime numbers to represent any even number (2n). That is, \( 2n=|(\text{N1N2N3...2a})+(\text{N1N2N3...2a})| \), whether \( a \) is an odd number when \( m \) is an even number, or \( a \) is an odd number when \( m \) is an odd number, 1 It can always be established the Goldbach conjecture can be proved.

5:55PM P03.00004 Reserves Classification and Well Pattern Infilling Adjustment in Tight Sandstone Gas Field – ZHI GUO1, AILIN JIA, Petrochina Research Institute of Petroleum Exploration and Development, TIGHT GAS DEVELOPMENT TEAM — Sulige is typical of tight sandstone gas field in China, with poor reservoir property and strong heterogeneity. The recovery factor is only about 30% under the current developing well pattern of 600m800m. Thus it is necessary to evaluate various types of reserves comprehensively and implement well pattern infilling adjustment respectively. Through fine reservoir description in dense well pattern and interference well test analysis, the reservoir distribution frequency was studied and reserves were classified into five types. Compared actual production data with modeling & simulation result, the relation of well pattern density, interference degree and recovery factor was researched for each type of reserves. Then, it can be concluded the appropriate well pattern density in various type of reserves area is 2~4 wells per square kilometers, and the ultimate recovery factor is about 50%. This research, makes reserves configuration clear in the tight sandstone gas field, provides geological basis for well pattern infilling adjustment in later development stage, lays a solid foundation for long-term stable production of gas field, and is also of certain reference significance to other gas field development under similar geological conditions.

1 No.1 research institute of petroleum in China
6:08PM P03.00005 Technologies advancement and prospect of natural gas development in China. AILIN JIA1, ZHI GUO, Petrochina Research Institute of Petroleum Exploration and Development, COMPLEX GAS RESERVOIR DEVELOPMENT TEAM — In recent years, international oil prices have stayed low and China strives to develop green energy. Under these backgrounds, natural gas has gradually become the key business of petroleum industry in China. After years of hardworking, several developmental technological breakthroughs are achieved for deep gas, tight gas, shale gas, coal-bed methane, development adjustment strategies, engineering technologies and decision-making system. With the depth of development, influenced by policy, environment and geological conditions, continuous efficient development of domestic natural gas faces many challenges: lower ratio of good quality reserves, more development expense, larger difficulties of remaining development benefit for unconventional gas reservoir, further upstream benefit compression, poorer stable production ability and fiercer competition in the energy market. In the future, natural gas development in China will enter into a new stage of emphasizing on both the unconventional gas and the conventional gas, gas imports will rise year by year, resulting in the increased external dependency, and natural gas will become the main growth engine in the process of energy structure adjustment.

1advanced research institute in China

6:21PM P03.00006 Plume-Chimney in Natural Convection from Large Sources. CHRISTOPHER CHU1, Universiti Malaysia Sabah — A natural draft chimney is defined by the existence of buoyancy and a solid wall barrier between two regions of fluids that differ in density. By preventing the mixing of two fluids at different densities in a fluid environment, there is effectively a wall-like characteristic, a chimney system without the solid walls. An “effective plume height” was pioneered by heat transfer engineers to close the solution loop of the natural convection performance of forced draft air-cooled heat exchangers, by modifying an atmospheric plume rise model. An improved prediction method was later developed and satisfactorily validated by both experiments and computational fluid dynamics (CFD) simulation of the natural convection performance of an industrial-scale air-cooled heat exchanger. The flow development region that has been identified in the field of jets is known as the zone of flow establishment (ZFE), but in lazy plumes of low Froude number it has stack effect. Two reasons for justifying the concept of plume-chimney are: 1) it affects the draft of a natural convection system of large source area by extending the solid-walled chimney height and 2) its existence anticipates a potential blockage to cross current such as wind in real natural convection systems, modifying the flow pattern.

The authors would like to offer their sincere thanks to the Ministry of Higher Education, Malaysia, for the kind assistance provided through fundamental grant No. FRG0022-TK-1/2006.

2nd Author is Md. Mizanur Rahman mizanur.rahman@mte.wub.edu.bd

Monday, November 25, 2019 5:16PM - 7:13PM — Session P04 Focus Session: Leidenfrost Drops and the Physics of the Vapor Layer 203 - Justin Burton, Emory University

5:16PM P04.00001 The vibrating life of Leidenfrost drops, AMBRE BOUILLANT1, LadHyX (Ecole polytechnique & CNRS) - PMMH (ESPCI Paris) — A volatile liquid is deposited on a hot solid can survive for minutes, which is known since Leidenfrost to rely on the presence of an insulating vapor layer beneath the liquid. In this levitating state, vapor prevents liquid adhesion leading to thriving dynamics, either spontaneous or forced. Among those spectacular features is the liquid ability to undergo a shape transition from axisymmetric to self-organized oscillations. Such liquid stars are not only encountered with Leidenfrost drops, but with weakly adhering liquids subjected to external periodic forcing. Brunet et al. reported stars are observed provided excitation exceeds a threshold. For Leidenfrost drops, however, the mechanism remains a mystery since there is no prescribed forcing - stars spontaneously appear and sustain. Various mechanisms were proposed invoking thermal convection. Yet, drops supported by steady air flow perform such oscillations. Our plan is to discuss the origin of Leidenfrost stars. We report that vapor self-vibrates, and explain the ensuing coupling that leads to those spectacular oscillations.

1Co-authors: Christophe Clanet and David Quere

5:29PM P04.00002 Characterizing the final moments of the Leidenfrost vapor layer, DANA HARVEY, JOSHUA MENDEZ, JUSTIN BURTON, Emory University — The vapor layer generated between a very hot surface and an evaporating liquid thins as the surface cools down. Eventually, hydrodynamic fluctuations of the liquid will cause the vapor layer to fail, leading to liquid/solid contact and explosive boiling. Using a new electrical technique we can characterize the failure of the vapor layer on sub-microsecond time scales. The vapor layer is treated as a complex circuit component with measurable impedance. A heated titanium electrode is lowered into a bath of salt water. A 10 MHz carrier signal passes through the vapor layer and the liquid so that the capacitive reactance of the system is low. The amplitude of the carrier wave is monitored in time. Using a model circuit for the vapor layer, physical properties of the layer can be determined. Within the first millisecond, a touchdown event occurs where the liquid rapidly wets the solid surface, characterized by a precipitous drop in the resistive part of the impedance. Then, the heat transfer to the liquid causes vaporization and boiling spreads from the point of contact. We expect this technique will also shed light on the sensitive role that salt plays in the Leidenfrost effect, as well as surface properties such as roughness.

1NSF DMR-1455086

5:42PM P04.00003 The nanoscale instability driving Leidenfrost film collapse, TOM Y. ZHAO, NEELESH A. PATANKAR, Northwestern University — Above a critical temperature known as the Leidenfrost point (LFP), a heated surface can suspend a liquid droplet above its own film of vapor. Here, we identify the vapor film instability for small length scales that ultimately sets the collapse condition at the LFP. From a linear stability analysis, it is shown that the main film stabilizing mechanisms are the liquid-vapor surface tension, viscous transport of vapor mass, and evaporation at the liquid-vapor interface. Meanwhile, van der Waals interaction between the bulk liquid and the solid substrate across the vapor phase drives film collapse. This physical insight into vapor film dynamics allows us to derive an ab-initio, mathematical expression for the Leidenfrost point of a fluid. The expression captures the experimental data on the LFP for different fluids under various surface wettabilities and ambient pressures. For fluids that wet the surface (small intrinsic contact angle), the expression can be simplified to a single, dimensionless number that encapsulates the nanoscale instability governing the LFP.
ANÁIS GAUTHIER, ESPCI Paris, France, GUILLAUME LAJOINIE, CHRISTIAN DIDDENS, DETLEF LOHSE, JACCO DROPLETS.

A miniature celestial system. The attraction between identical menisci impacts the motion of approaching droplets which spontaneously orbit around each other – mirroring what happens on a droplet-laden cryogenic bath. In such freezing conditions, the droplets (which do not evaporate) can levitate for a dozen of minutes. We show here how the deformability of the liquid substrate dramatically impacts the Leidenfrost dynamics. First, we show that a micrometer-sized instability grows within the film sustaining the drop, which causes a partial redirection of the vapor flow and generates spontaneous self-propulsion. The drops then behave as active particles, which hover in straight lines above the bath and form a remarkably regular pattern. In addition, the bath surface is deformed at the millimeter scale by the droplets' weight. Due to this non-wetting meniscus, the particles are repelled at large distance by objects dipped into the bath: this can be used as a contactless method to finely control the particles trajectories. Conversely, we show that the attraction between identical menisci impacts the motion of approaching droplets which spontaneously orbit around each other – mirroring a miniature celestial system.

Leidenfrost droplets. ANÁIS GAUTHIER, ESPCI Paris, France, GUILLAUME LAJOINIE, CHRISTIAN DIDDENS, DETLEF LOHSE, JACCO SNOEIJER, DEVARAJ VAN DER MEER, PoF - University of Twente. The Netherlands — Ambient temperature drops deposited on a liquid nitrogen bath can be maintained in the inverse Leidenfrost state, a levitating state that is enabled by a continuous vapor flow produced by the cryogenic bath. In such freezing conditions, the droplets (which do not evaporate) can levitate for a dozen of minutes. We show here how the deformability of the liquid substrate dramatically impacts the Leidenfrost dynamics. First, we show that a micrometer-sized instability grows within the film sustaining the drop, which causes a partial redirection of the vapor flow and generates spontaneous self-propulsion. The drops then behave as active particles, which hover in straight lines above the bath and form a remarkably regular pattern. In addition, the bath surface is deformed at the millimeter scale by the droplets' weight. Due to this non-wetting meniscus, the particles are repelled at large distance by objects dipped into the bath: this can be used as a contactless method to finely control the particles trajectories. Conversely, we show that the attraction between identical menisci impacts the motion of approaching droplets which spontaneously orbit around each other – mirroring a miniature celestial system.

Leidenfrost droplets on superheated substrates. TAMAL ROY, Postdoctoral Research Associate, UDDALOK SEN, Postdoctoral Researcher, RANJAN GANGULY, Professor, LOUIS A. ANGELONI, Graduate Assistant, W. ANDREAS SCHROEDER, CONSTANTINE M. MEGARIDIS, Professor, DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING, UNIVERSITY OF ILLINOIS AT CHICAGO, USA TEAM, DEPARTMENT OF POWER ENGINEERING, JADAVPUR UNIVERSITY, KOLKATA, INDIA TEAM, DEPARTMENT OF PHYSICS, UNIVERSITY OF ILLINOIS AT CHICAGO, USA TEAM — Droplet impact on superheated smooth solids has attracted enormous attention since Leidenfrost reported the eponymous phenomenon more than 250 years ago. The related literature is almost exclusively focused on single component liquids, which maintain their identity even under disruptive boiling conditions. In this work, we provide evidence for the existence of a new regime - termed explosive boiling - for the impact of bicomponent (ethanol and water) droplets on superheated substrates at temperatures between the respective Leidenfrost temperatures of the two liquid constituents. This regime is characterized by a violent shattering of the main droplet upon impact, and is observed only for a certain range of alcohol concentrations of the binary mixture. We explore this behavior experimentally through high-speed imaging at different substrate temperatures, droplet concentrations, and impact velocities. Furthermore, we provide interferometric evidence for the cause of occurrence of this unprecedented regime.

6:08PM P04.00005 ABSTRACT WITHDRAWN

6:21PM P04.00006 Self-propulsion and capillary orbits of inverse Leidenfrost droplets. ANÁIS GAUTHIER, ESPCI Paris, France, GUILLAUME LAJOINIE, CHRISTIAN DIDDENS, DETLEF LOHSE, JACCO SNOEIJER, DEVARAJ VAN DER MEER, PoF - University of Twente. The Netherlands — Ambient temperature drops deposited on a liquid nitrogen bath can be maintained in the inverse Leidenfrost state, a levitating state that is enabled by a continuous vapor flow produced by the cryogenic bath. In such freezing conditions, the droplets (which do not evaporate) can levitate for a dozen of minutes. We show here how the deformability of the liquid substrate dramatically impacts the Leidenfrost dynamics. First, we show that a micrometer-sized instability grows within the film sustaining the drop, which causes a partial redirection of the vapor flow and generates spontaneous self-propulsion. The drops then behave as active particles, which hover in straight lines above the bath and form a remarkably regular pattern. In addition, the bath surface is deformed at the millimeter scale by the droplets' weight. Due to this non-wetting meniscus, the particles are repelled at large distance by objects dipped into the bath: this can be used as a contactless method to finely control the particles trajectories. Conversely, we show that the attraction between identical menisci impacts the motion of approaching droplets which spontaneously orbit around each other – mirroring a miniature celestial system.

6:34PM P04.00007 Explosive behavior of binary drops upon impact on a hot solid. PIERRE CHANTELOT, PALLAV KANT, JULIETTE COLIN, Physics of Fluids group, University of Twente, ANDREA PROSPERETTI, Physics of Fluids group, University of Twente and Department of Mechanical Engineering, University of Houston, DETLEF LOHSE, Physics of Fluids group, University of Twente and Max Planck Institute for Dynamics and Self-Organization — The vapor layer generated during the impact of a drop on a hot plate can prevent contact between the liquid and solid. The minimum temperature, $T_\text{min}$, needed to observe this dynamic Leidenfrost effect depends on the impact speed, and the thermal properties of the solid and liquid. Here, we perform impacts of binary drops of miscible liquids with different boiling points while monitoring the existence of contact using Total Internal Reflection (TIR) imaging. We report the occurrence of explosions, associated with a sudden failure of the vapor film, at the transition from levitation to contact. We study the destabilization of the vapor layer by measuring its dynamics for different liquid combinations and mixing ratios.

6:47PM P04.00008 Low-g Demonstrations of Leidenfrost Droplet Impacts with Applications to Non-Contact Fluidics Processing in Space. RAWAND RASHEED, MARK WEISLOGEL, Portland State University — Leidenfrost phenomenon has been studied extensively for its role in applications ranging from nuclear reactor cooling, to metals manufacturing, combustion, and other fields. Herein, Leidenfrost phenomenon is exploited as a potential solution to spacecraft water processing challenges by providing a method for non-contact fluid distillation. Leidenfrost investigations have been almost exclusively conducted in terrestrial environments and are in turn largely defined by the ever-presence of gravity. In this work we demonstrate a variety of dynamic Leidenfrost effects for large liquid droplets in the microgravity environment of a 2.1 second drop tower. A scaling model for the impact reveals the ease with which ‘non-contact’ impacts are achieved at low Weber numbers and low gravity levels. We find the vapor film thicknesses at impact are sub-millimetric as estimated analytically and confirmed qualitatively via experiments. Dynamic drop impact experiments are extended to a variety of heated substrates including macro-pillar arrays, confined passageways, and others. Leidenfrost droplet evaporation rates are measured analytically and confirmed qualitatively for binary drops of miscible liquids with different boiling points while monitoring the existence of contact using Total Internal Reflection (TIR) imaging. We report the occurrence of explosions, associated with a sudden failure of the vapor film, at the transition from levitation to contact. We study the destabilization of the vapor layer by measuring its dynamics for different liquid combinations and mixing ratios.

7:00PM P04.00009 Unexpected Suppression of Leidenfrost Phenomenon on Superhydrophobic Surfaces. MENG SHI, RATUL DAS, SANKARA ARUNACHALAM, HIMANSHU MISHRA, King Abdullah University of Science and Technology — The Leidenfrost phenomenon ascribes to a non-equilibrium situation, wherein a liquid droplet levitates above a superheated surface. Superhydrophobic surfaces are believed to dramatically reduce water’s Leidenfrost point (LFP - the temperature when the Leidenfrost phenomenon occurs), even approaching its boiling point in some cases. The causation is that superhydrophobic surfaces robustly entrap air when brought in contact with water, which reduce the contact area and adhesion between the droplet and the surface, and thus promote the formation of the vapor layer on heating. Here, we report on a curious exception. Using high-speed imaging, we investigated water droplets placed on hot superhydrophobic SiO2/Si surfaces adorned with arrays of doubly reentrant pillars (DRPs). We found the LFP for water droplets on SiO2/Si DRPs could be significantly higher than that smooth SiO2/Si surfaces, even though the former exhibits superhydrophobicity and the latter is hydrophilic. Thus, we advance the notion that superhydrophobic surfaces may not always lower the LFP of water. We will present deeper insights into our observations based on complementary experiments and theory.

Monday, November 25, 2019 5:16PM - 7:13PM – Session P05 Turbulent Combustion Modeling and Methods 204 - James Brasseur, University of Colorado, Boulder
5:16PM P05.00001 LES/PDF of Sandia flame D using a coupled adaptive chemistry and tabulation approach1, ASHISH NEWALE, YOUWEN LIANG, STEPHEN POPE, PERRINE PEPIOT, Cornell University — LES/PDF methods are known to provide accurate results for challenging turbulent combustion configurations. This higher level of fidelity, however, comes at the cost of added computational expense compared to other state-of-the-art methods. To reduce the magnitude of this cost differential, the majority of LES/PDF computations performed to date have used reduced mechanisms. We have recently proposed a coupled adaptive chemistry and tabulation approach to enable the use of the detailed mechanisms in LES/PDF computations. Specifically, we proposed a coupled pre-partitioned adaptive chemistry (PPAC) and in-situ adaptive tabulation (ISAT) method (Newale et al., CTM 2019). The proposed coupled method showed encouraging results in a partially stirred reactor configuration. In this work, we examine the performance of the coupled PPAC-ISAT method in a LES/PDF computation of Sandia flame D. We demonstrate that the coupled technique enables the use of detailed mechanisms at a significantly reduced computational cost, while retaining the level of fidelity attained by using the detailed mechanism without approximations.

1U.S. Department of Energy Office of Science under Award Number DE-FG02-90ER14128

5:29PM P05.00002 In-Situ Adaptive Manifolds: Enabling simulations of complex turbulent reacting flows, CRISTIAN E. LACEY, ALEX G. NOVOSELOV, MICHAEL E. MUELLER, Princeton University — Reduced-order manifold approaches to turbulent combustion modeling traditionally involve precomputation of manifold solutions and tabulation of the thermochemical database versus a small number of manifold variables. However, additional manifold variables are required as the complexity of turbulent combustion processes increases, for example, multi-modal combustion or combustion featuring multiple and/or inhomogeneous inlets. This increase in the number of manifold variables comes with an increase in the computational cost of precomputing a greater number of manifold solutions, most of which are never actually utilized in a CFD calculation. The memory required to store the pretabulated high-dimensional thermochemical database also increases, limiting the complexity of reduced-order manifold combustion models. In this work, a new In-Situ Adaptive Manifolds (ISAM) approach is developed that overcomes this limitation by combining ‘on-the-fly’ calculation of manifold solutions with In-Situ Adaptive Tabulation (ISAT), enabling the use of more complex manifold-based turbulent combustion models. The performance of ISAM is evaluated via LES calculations of canonical turbulent flames.

5:42PM P05.00003 Fundamental Differences between Large-Eddy Simulation of Incompressible Turbulence vs Premixed Turbulent Combustion, JAMES BRASSEUR, Univ of Colorado, YASH SHAW, Penn State University, PAULO PAES, Gamma Technologies, YUAN XUAN, Penn State University — In contrast with RANS where the modeled terms are of leading order, the LES framework requires that the modeled subfilter-scale (SFS) contributions be of lower order than the leading-order terms. This will be the case if the resolved-scale (RS) contributions to the triadic sum of advective nonlinearities in spectral space dominate the SFS triads, requiring an effective grid that resolves well Reynolds stress motions. Turbulent combustion deviates from the LES framework is several key ways, primarily in the existence of the chemical source terms that lead to the release of thermal energy at scales generally unresolved. In this study we quantify the dominant SFS contributions to the key nonlinearities that underlie RS evolution in LES of premixed turbulent combustion to isolate fundamental deviations from the LES framework. With a new method to remove spurious spectral content in inhomogeneous directions, we apply a concurrent physical-Fourier space methodology to compressible DNS of flame-turbulence interactions to isolate the triadic structure of advective nonlinearities and the quadratic structure of chemical nonlinearities in the Fourier representation from which the dynamically dominant contributions are determined. We find that when the RS fluctuations in momentum and scalar dissipation rate. This methodology allows for a clear assessment of the errors that arise from the three aspects of a model for combustion: turbulence-closure, modeling the turbulence-chemistry interaction, and chemical kinetics closures. As there is no conflation of errors arising via the chemical kinetics, the data can be used to directly assess the performance of RANS closures in representing the turbulence and the turbulence-chemistry interaction. Conversely, by comparing our DNS with one closed with a higher fidelity chemistry model from Attili and co-authors, any discrepancies can be attributed to the different chemical kinetics closures. Results are shown for the k−ε model with presumed-PDF approach for a non-premixed n-heptane flame.

5:55PM P05.00004 Direct Numerical Simulation of the Temporally-Evolving Reacting Jet for Model Error Assessment of RANS-based Closures for Non-Premixed Turbulent Combustion, BRYAN REUTER, TODD OLIVER, ROBERT MOSER, Oden Institute for Computational Engineering and Sciences, UT-Austin — High-fidelity DNS data of the temporal reacting jet is generated using a laminar flamelet closure, the same chemistry model as is typically employed in RANS-based approaches. The simulations solve the low-Mach Navier-Stokes equations with a pseudospectral Fourier-Galerkin/B-Spline collocation approach and a novel second-order, explicit time marching scheme. The turbulence is fully resolved, but the chemistry is modeled by obtaining the density from a flamelet library which is a function of the local mixture fraction and scalar dissipation rate. This methodology allows for a clear assessment of the errors that arise from the three aspects of a model for combustion: turbulence closures, modeling the turbulence-chemistry interaction, and chemical kinetics closures. As there is no conflation of errors arising via the chemical kinetics, the data can be used to directly assess the performance of RANS closures in representing the turbulence and the turbulence-chemistry interaction. Conversely, by comparing our DNS with one closed with a higher fidelity chemistry model from Attili and co-authors, any discrepancies can be attributed to the different chemical kinetics closures. Results are shown for the k−ε model with presumed-PDF approach for a non-premixed n-heptane flame.

6:08PM P05.00005 Timescale Analysis Procedure Applied to Premixed and Non-Premixed Turbulent Combustion1, SALVADOR BADILLO-RIOS, University of California, Los Angeles, MATTHEW HARRISON, VENKATESWARAN SANKARAN, AFRL Research Laboratory, Rocket Propulsion Division, ANN KARAGOZIAN, University of California, Los Angeles — Full detailed kinetics in turbulent combustion simulations can be computationally prohibitive for propulsion systems. While reduced kinetic models are often selected to reduce cost, there needs to be a clear understanding of their impact on the phenomena being modeled. The present study examines the effects of alternative kinetic models and flow parameters on turbulent combustion processes associated with non-premixed combustion in a shear-coaxial rocket injection configuration, as a means of determining the conditions under which turbulent reaction phenomena may be altered via the kinetics. A combination of 2D axisymmetric parametric studies and selected 3D simulations for a single element shear coaxial rocket injector are performed, incorporating GRI-Mech 3.0 and several alternative reduced kinetic models representing the combustion of gaseous methane and oxygen. Systematic timescale analysis procedures, e.g., the Chemical Explosive Mode Analysis (CEMA) procedure based on the Jacobian matrix of the chemical source term, are applied to explain differences in observed flame behaviors. This approach can serve as a quantitative method for the systematic detection of critical flame features, species, and reactions.

1Supported by NDSEG, ERC, and AFOSR (PO: Dr. Chiping Li)
6:21PM P05.00006 Large Eddy Simulations of turbulent flames using two-dimensional reduced-order manifold models

ALEX G. NOVOSELOV, CRISTIAN E. LACEY, MICHAEL E. MUELLER, Princeton University — Recently, we have developed a set of two-dimensional manifold equations describing multi-modal combustion using the mixture fraction and a generalized progress variable. Information about the underlying mode of combustion is encoded in three scalar dissipation rates that appear as parameters in the two-dimensional equations. In this work, Large Eddy Simulations (LES) of turbulent hydrogen flames in both the asymptotic limits of combustion (i.e. nonpremixed and premixed) and multi-modal flames are performed using this new turbulent combustion model. These simulations are compared to LES performed using traditional one-dimensional manifold-based turbulent combustion models as well as experimental measurements. The new model is shown to describe the flames in the asymptotic limits just as well as the one-dimensional models but without any a priori knowledge of the flame necessary to select the model. Additionally, the new model is shown to be able to describe the behavior of multi-modal turbulent flames where traditional one-dimensional models fail.

6:34PM P05.00007 A Dispersion Model for Turbulent, Multi-Component Reacting Flows

OMKAR SHENDE, ALI MANI, Stanford University — It is theoretically and computationally challenging to build reduced-order models for turbulent reacting flows as the underlying chemical and transport processes are individually complex. Furthermore, an understanding of the coupled effects of these phenomena remains elusive. However, deeper insight into turbulent transport effects on reaction dynamics is essential for the future design of efficient energy systems. Using theory developed for non-reactive dispersion of scalars and linear reactions, an algebraic Reynolds-averaged Navier-Stokes model for capturing unresolved interactions between multi-component scalar reactions in turbulent flows is developed. This work extends the modified gradient diffusion model by Corrsin (JFM, vol. 11, p.407-416) beyond single-component transport phenomena with linear reactions. Using two- and three-dimensional direct numerical simulations, it is shown that this model improves prediction of mean quantities compared to traditional results.

1This work was supported by the Stanford Graduate Fellowship and the National Science Foundation GRFP

6:47PM P05.00008 Conditional Reynolds stress modeling in turbulent premixed jet flames

JINYOUNG LEE, MICHAEL MUELLER, Princeton University — Conventional turbulence modeling approaches based on traditional unconditional averaging implicitly assume that combustion heat release does not affect turbulence. However, in turbulent premixed flames at low Karlovitz number, combustion-induced dilatation and flame motion significantly modify turbulence. Instead of relying on unconditional averaging, simultaneously solving momentum and scalar transport equations conditionally averaged on a flame structure variable could provide a superior framework for modeling combustion-affected turbulence since the flame dynamics are embedded into the flame-conditioning. The primary challenge in this approach is developing closure models for conditional terms, which evolve in both physical and conditional spaces. In this work, a new model for conditional Reynolds stresses, which appear in the conditionally averaged momentum equations, has been developed. The new model consists of a conditional Boussinesq-like term and a new term representing turbulent momentum transport in conditional space. A theoretical scaling of the new model has been investigated, and the model performance has been evaluated in a priori analyses using DNS databases of turbulent premixed jet flames at low and high Karlovitz numbers.

7:00PM P05.00009 DNS analysis of flame propagation for systematic variations in turbulence scales and intensity

SHREY TRIVEDI, R. STEWART CANT, University of Cambridge — Direct Numerical Simulation (DNS) of premixed hydrocarbon flames is performed using the DNS code Senga2 to investigate the role of varying integral length scale $l_0$ as well as the turbulence intensity $u'$ in the thin reaction zone regime. Different cases are studied by either systematically varying $l_0$ at constant $u'$ or by systematically varying $u'$ at constant $l_0$. Several aspects of these flames are compared. The turbulent flame speed $s_T$ is found to decrease as $l_0$ decreases or when $u'$ increases. The ratio of $s_T/l_0$ is generally accounted for by the area change $A_f/A_b$, but a significant deviation is observed between these values at high $u'$ cases. The size of the smallest scales of turbulence also decreases with $l_0$ and the interaction of these small scales with the flame produces an increase in the mean curvature of the flame which eventually reaches a saturation point. There is also an increase in the number of flame-flame interactions with decreasing $l_0$. Individual flame-flame interaction topologies are identified and analysed. All these findings serve to further our understanding of turbulent combustion and the role that the turbulence length scales play in flame propagation.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P06 Thermoacoustic Instability

5:16PM P06.00001 Three-dimensional Scanning Laser Induced Fluorescence for the investigation of flame dynamics on annular combustor instabilities

DIRREN GOVENDER, SAMUEL WISEMAN, JAMES R DAWSON, NICHOLAS WORTH, Department of Energy and Process Engineering, Norwegian University of Science and Technology, N-7491 Trondheim, Norway — The need for three dimensional techniques is of importance when investigating turbulent asymmetrical flame structures. In combustion systems, self-excited thermo-acoustic instabilities are a prevalent problem and many studies are aimed at better understanding the phenomena. In this study, a Scanning OH* Laser Induced Fluorescence method is introduced to spatially resolve three dimensional flame structures during forced azimuthal oscillations in an annular combustor. The scanning setup consisted of a galvanometer mirror that sweeps a laser sheet through the flame while imaging at 10kHz. The obtained images were phase averaged based on the forcing cycle and the results were used to reconstruct the three dimensional flame structure. A variety of forced standing and spinning modes were investigated. Phase-averaged Flame Surface density (FSD) was used to characterize the three-dimensional flames. A series of calibrations were performed and evaluated for the transformation of image space to object space. This consisted of an additional camera to characterize the laser sheet positions and track any movements of the calibration plate while calibrating the in-plane camera. The results shows the scanning method’s ability to resolve three-dimensional flame structures and dynamics.

5:29PM P06.00002 Exploration of Unstable Spray Flames Using Reduced-Order Models

ABDULLA GHANI, THOMAS STEINBACHER, ALP ALBAYRAK, WOLFGANG POLIFKE, TU Munich — We investigate low-frequency thermoacoustic instabilities of a swirling spray flame. This study is based on an aeronautical lab-scale experiment, for which a reduced-order model (ROM) has been generated. A parametric study of the ROM suggests that the instability mechanism is caused by an intrinsic thermoacoustic (ITA) feedback loop. Further analysis such as separation of acoustic and ITA modes or the scaling of the ITA frequency with the bulk velocity confirm the ITA feedback loop as the instability driver. Results of the ROM agree well with experimental observations and demonstrate the effectiveness of ROMs.
5:42PM P06.00003 Effect of Global Acceleration on the Stability of a Thermocoustically Coupled Shear Layer in a Backward-Facing Step Combustor, JOEL VASANTH, SATYANARAYANAN CHAKRAVARTHY, Department of Aerospace Engineering, IIT Madras — The impact of the acoustic field on variable density flow in a step combustor is explored via a linear stability analysis of the low Mach number Navier-Stokes equations. In the equi-time scales limit, the acoustic feedback to the flow reduces to a global acceleration (GA) as a momentum source. The velocity profiles are parametrized by shear layer thickness δ and reverse flow ratio β. To close the equation set, the n−r flame model is used. A local spatio-temporal perturbation analysis shows a shift in the absolutely/convectively unstable (AU/CU) transition boundaries towards the AU zone in the β-δ space as the flame response gain is increased. Further, the AU modes are less responsive to the acoustic feedback than the CU modes, affirming the semi-open/fully-closed loop mechanisms of acoustic feedback advanced in recent literature. Competing contributions of the vorticity equation source terms to the overall vorticity generation show that the GA is always destabilizing unlike the density gradient, which could be stabilizing under some conditions. The GA and the resultant of vorticity production and baroclinic vorticity are in phase under all conditions, which implies the forcing nature of the GA that unconditionally promotes combustion instability.

5:55PM P06.00004 Intrinsic thermoacoustic modes in systems with two axially separated heat sources, PHILIP BUSCHMANN, JONAS MOECK, NTNU — Modern gas turbines for power generation are required to provide high operational flexibility while complying with emission limits. Axially staged flames or sequential combustors are advantageous for this purpose. However, the distributed heat release rate in these systems gives rise to new thermoacoustic interaction phenomena. Recently, it has been shown that the forcing nature of the GA that unconditionally promotes combustion instability.

6:08PM P06.00005 Large-eddy simulations of a heated and combusting Rijke tube configuration, WEI XIAN LIM, WAI LEE CHAN, Nanyang Technological University, Singapore — Due to its potential to cause catastrophic damages to combustion engines, thermoacoustic instability is an active area of research that demands further understanding. The process to experimentally identify or eliminate thermoacoustic instability effects can be costly, so the ability to employ numerical simulation techniques will be advantageous. In this work, a canonical thermoacoustic configuration, known as the Rijke tube, was chosen to study the coupling effects between the heat-addition and acoustic pressure fluctuations. To this end, two large-eddy simulation cases were studied, one with heating wire and the other with combustion represented by the flamelet/progress-variable model. In all simulations, proper boundary coupling effects between the heat-addition and acoustic pressure fluctuations. To this end, two large-eddy simulation cases were studied, one with heating wire and the other with combustion represented by the flamelet/progress-variable model. In all simulations, proper boundary conditions and buoyancy force were accounted for to represent the actual conditions of Rijke tube in a laboratory space. Good results were obtained for the heating-wire case when comparing the theoretical and numerical resonant frequency. Meanwhile, additional flame-acoustics interactions were also observed in the flamelet/progress-variable case.

6:21PM P06.00006 Receptivity of a transversely forced jet\textsuperscript{1}, JOSE G. AGUILAR, EIRIK ASOY, JAMES DAWSON, NICHOLAS WORTH, Institute for Energy and Process Engineering, Norwegian University of Science and Technology (NTNU) — Annular combustion chambers are often used in gas turbines. These configurations are prone to thermoacoustic instabilities which emerge as standing or spinning azimuthal modes among others. During the instability the flow conditions of typical flames resemble swirling annular jets which are transversely forced by an acoustic wave. Previous studies suggest that in acoustically compact configurations the flame response to transverse flow excitation is strong if it is seen locally (within sections of the flame), but it is weak if it is seen globally given that there is negligible contribution to the total fluctuations of the heat release. By means of experimental analysis and numerical simulation the present study aims to quantify the receptivity of a more simplistic flow configuration, an isothermal jet subject to transverse forcing and elucidate the mechanism for its strong receptivity with a steady jet at different Mach numbers. The experiments show that the response of the jet depends on the position of the acoustic wave and is described by a mixture of the symmetric and antisymmetric modes. Receptivity analysis is carried out using numerical simulation in order to quantify the strength of each mode at the different forcing positions.

\textsuperscript{1}NCCS

6:34PM P06.00007 Structure of thermoacoustic modes in can-annular combustion systems, JONAS MOECK, NORWEGIAN UNIVERSITY OF SCIENCE AND TECHNOLOGY, GIULIO GHIRARDO, MIRKO BOTHEN, Ansaldo Energia Switzerland — Large gas turbines for power generation employ a can-annular combustor architecture. In contrast to annular combustors, in a can-annular system, the flames are isolated and burn in individual cans; however, there is acoustic communication between adjacent cans at the turbine inlet through a small gap. The thermoacoustic modal structure of a can-annular combustor significantly differs from that of the extensively studied annular combustor. We consider a can-annular configuration as a system composed of weakly coupled, nominally identical subcomponents. This provides a consistent framework for studying the spectral properties and explaining prevalent phenomena in these systems, such as eigenvalue clustering, and mode localization when the symmetry is perturbed. Based on a generic can-annular model system, we illustrate the general modal structure and derive the equivalent downstream boundary condition that links the multi-can resonances to a single-can model.

6:47PM P06.00008 Time-Accurate Calibration of a Thermoacoustic Model on Experimental Images of a Forced Premixed Flame, HANS YU, USHnish SENGUPTA, MATTHEW JUNIPER, Department of Engineering, University of Cambridge, LUCA MAGRI, Department of Engineering, University of Cambridge; Institute for Advanced Study, Technical University of Munich — Thermoacoustic instabilities are a persistent challenge in the design of jet and rocket engines. The time-accurate calculation of thermoacoustic instabilities is challenging due to the presence of both aleatoric and epistemic uncertainties, as well as the extreme sensitivity to small changes in certain parameters. We extend our previous work (Yu et al., CTR summer program 2018; Yu et al., J. Eng. Gas Turbines Power 2019) by applying our recently published level-set data assimilation framework (Yu et al., J. Comput. Phys. 2019) to experimental images of a forced premixed flame. We force a Bunsen flame with a loudspeaker and record videos at different frequencies and amplitudes. Data assimilation provides an optimal estimate of the true state of a system, and improves the predicted shape and location of the flame. Parameter estimation uses the data to find a maximum-likelihood set of parameters for the model while simultaneously quantifying their uncertainty and identifying deficiencies in the model. We demonstrate our level-set data assimilation framework using both the ensemble Kalman filter and smoother. More generally, we take a physics-informed, reduced-order model and use statistical learning techniques to make it quantitatively accurate.
7:00PM P06.00009 Forced and mutual synchronization of periodic and aperiodic oscillations in a self-excited thermoacoustic system\textsuperscript{1}, YU GUAN, The Hong Kong University of Science and Technology, VIKRANT GUPTA, Southern University of Science and Technology, LARRY K.B. LI, The Hong Kong University of Science and Technology — Recent studies have shown that external forcing is effective in controlling both periodic and aperiodic thermoacoustic oscillations. We examine the mutual synchronization of pressure ($p'$) and heat-release-rate ($q'$) fluctuations in a prototypical thermoacoustic system undergoing forced synchronization by periodic acoustic forcing. When unforced, the system can oscillate periodically, quasiperiodically or chaotically. For all three types of oscillations, we find several common features, including (i) the presence of a $T^2$ quasiperiodic state before lock-in in which asynchronous quenching occurs without mutual synchronization between $p'$ and $q'$, (ii) the emergence of synchronization between $p'$ and $q'$ and the forcing signal at lock-in, and (iii) the destruction of synchronization beyond lock-in. As well as providing new insight into the way external forcing affects the mutual synchronization of $p'$ and $q'$, this study shows that, regardless of its initial unforced state, a thermoacoustic system synchronized to external forcing does not necessarily remain synchronized if the forcing becomes too strong. For effective control, this implies that the forcing amplitude should be limited to a value just sufficient to cause the onset of lock-in.

\textsuperscript{1}This work was funded by the Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815).

Monday, November 25, 2019 5:16PM - 7:00PM — Session P07 Non-Linear Dynamics and Chaos II

5:16PM P07.00001 Passive and active mixing in vortex chain flows\textsuperscript{1}, TOM SOLOMON, Bucknell University — In 1988\textsuperscript{2} Jerry Gollub and I published an article that first identified chaotic mixing in an oscillating chain of alternating vortices, a model developed to describe transport in time-dependent Rayleigh- Bénard convection. In the thirty years since, the vortex chain flow (and the similar double-gyre flow) have become paradigm flows for studying fluid mixing in closed flows. In this talk, I’ll review studies that have demonstrated both enhanced diffusion and suppression of vortex stretching in this system, as well as extensions to vortex arrays and three-dimensional, time-independent flows, which also show chaotic mixing. We will also discuss recent studies of active mixing – both of propagating reaction fronts and of swimming microbes – in vortex chain flows.

\textsuperscript{1}Supported by NSF Grants DMR-1361881 and DMR-1806355.


5:29PM P07.00002 Particle-scale fluctuations and hindered settling of a granular dispersion at low-Re\textsuperscript{1}, TED BRZINSKI, JAMES STADLER, IVAN TSEYTLIN, CHARLES WALKER, Haverford College — Dense dispersions of grains sedimenting in a fluid at low-Re are characterized by mean settling velocities which are hindered relative to Stokes settling. Hindered settling data can be collapsed to a master curve which is well-described by the Richardson-Zaki function $H(\phi) \equiv v/v_s = (1-\phi)^{\alpha}$, where $\phi$ is the particle volume fraction, but with different exponents: $\alpha \approx 5.5$ for systems with a small Peclet number, and $\alpha \approx 4.5$ for systems with large Peclet number. This branching occurs at a surprisingly large value of Peclet number. We report the results of our latest experimental investigations into this unexpected behavior. Using diffusing-wave spectroscopy, we characterize the spatio-temporal particle velocity fluctuations on scales as small as the expected Brownian motion for our experimental systems, and systematically vary Peclet number around the branching-point to quantify differences in the grain-scale dynamics between systems on either branch of the hindered settling function. We also present a model for the self-diffusion of the particles below the transition and increases linearly with strain amplitude above. For a 40% volume fraction suspension, we see low-q long-wavelength fluctuations suppressed near the critical strain of $\gamma \approx 0.9$ with $S(q) \propto \gamma^{1/2}$.

\textsuperscript{1}Acknowledgment is made to the donors of The American Chemical Society Petroleum Research Fund for support of this research.

5:42PM P07.00003 Random Organization: A Gift that Keeps on Giving\textsuperscript{1}, DAVID PINE, SANUEL WILKIN, PAUL CHAIKIN, NYU — Non-Brownian suspensions of spheres periodically sheared at low Reynolds number explore new configurations through collisions in an otherwise reversible flow. Below a critical strain, the particles remain active until they find a configuration with no collisions and fall into an absorbing state. Simulations by Hexner & Levine show that the system becomes hyperuniform near the critical configurations through collisions in an otherwise reversible flow. Below a critical strain, the particles remain active until they find a configuration with no collisions and fall into an absorbing state. This study shows that, regardless of its initial unforced state, a thermoacoustic system synchronized to external forcing does not necessarily remain synchronized if the forcing becomes too strong. For effective control, this implies that the forcing amplitude should be limited to a value just sufficient to cause the onset of lock-in.

\textsuperscript{1}This work is supported by the MRSEC Program of the National Science Foundation under Award Number DMR-1420073.

5:55PM P07.00004 Nonlinear Dynamics in Sports, ANETTE HOSOI, Massachusetts Institute of Technology — In celebration of Jerry Gollub’s remarkable contributions to nonlinear dynamics, I would like to discuss a few examples of complex dynamical behavior in sports. In free-flowing team sports such as soccer, basketball, and hockey, the relationships between individual athletes can give rise to collective behavior that can enhance (or diminish) the effectiveness of the team. New high-quality tracking data of professional basketball and soccer players, i.e. center of mass coordinates at 25Hz with centimeter-scale resolution, reveals signatures of player fatigue, decision-making aptitude, and other athlete traits that have historically been unquantifiable. Here we borrow concepts from the analysis of the collective motion of birds and fish, e.g. schooling and flocking, to begin to draw insights from tracking data and construct a low dimensional portrait of athlete characteristics.

6:08PM P07.00005 Elastic Alfven waves in elastic turbulence, VICTOR STEINBERG, Weizmann Institute of Science, ATUL VARSHNEY, Institute of Science and Technology Austria, Am Campus 1, 3400 Klosterneuburg, Austria — Speed of sound waves in gases and liquids are governed by the compressibility of the medium. There exists another type of non-dispersive wave where that has speed depends on stress instead of elasticity of the medium. A well-known example is the Alfven wave, which propagates through plasma permeated by a magnetic field with the speed determined by magnetic tension. An elastic analogue of Alfven waves has been predicted in a flow of dilute polymer solution where the elastic stress of the stretching polymers determines the elastic wave speed. Here we present quantitative evidence of elastic Alfven waves in elastic turbulence of a viscoelastic creeping flow between two obstacles in channel flow. The key finding in the experimental proof is a nonlinear dependence of the elastic wave speed $c_{el}$ on the Weissenberg number $W_i$, which deviates from predictions based on a model of linear polymer elasticity.
6:21PM P07.00006 Mixing in Turbulent Fluids: From Quasi Two-Dimensional Systems to Stably-Stratified Shear Flows, ROBERT ECKE, Los Alamos National Laboratory — Mixing in fluids represents one of the most important roles of chaotic or turbulent flows, being responsible for, among many things, the dispersion of pollutants, the efficient mixing of fuel/gas in internal combustion engines, and the distribution of heat and salinity in the ocean. I will discuss Lagrangian and Eulerian measurements of a thin layer driven by electromagnetic forcing which is an experimental realization of two-dimensional turbulence. I will also present experimental results of fully three-dimensional stably-stratified shear flows that represent geophysical problems arising from, for example, oceanic overflows and river estuaries. The simultaneous planar velocity and density fields are determined using particle image velocimetry and planar laser-induced fluorescence. Different analysis methods for characterizing the interplay between mixing and turbulence are explored including Lagrangian methods obtained from high-resolution particle tracking, filtering (coarse-graining) applied to both systems, and comparison with traditional turbulence analysis approaches. These different analysis techniques give physical insight into the mechanisms of mixing and transport in scalar advection, both the passive and active cases.

6:34PM P07.00007 Geometric phase and dimensionality reduction in locomoting living systems, JENNIFER RIESER, Georgia Institute of Technology, CHAOHUI GONG, Bito Robotics, HENRY ASTLEY, University of Akron, PERRIN SCHIEBEL, Georgia Institute of Technology, ROSS HATTON, Oregon State University, HOWIE CHOSE, Carnegie Mellon University, DANIEL GOLDMAN, Georgia Institute of Technology — The apparent ease with which animals move requires the coordination of their many degrees of freedom to manage and properly utilize environmental interactions. Identifying effective strategies for locomotion has proven challenging, often requiring detailed models that generalize poorly across modes of locomotion, body morphologies, and environments. We present the first biological application of a gauge-theory-based geometric framework for movement, originally proposed by Wilczek and Shapere nearly 40 years ago, to describe self-deformation-driven movements through dissipative environments. Using granular resistive force theory to model environmental forces and principal components analysis to identify a low-dimensional space of animal postures and dynamics, we show that our approach captures key features of how a variety of animals, from undulatory swimmers and slitherers to sidewinding rattlesnakes, coordinate body movements and leverage environmental interactions to generate locomotion. Our results demonstrate that this geometric approach is a powerful and general framework that enables the discovery of effective control strategies, which could be further augmented by physiologically-relevant parameters and constraints to provide a deeper understanding of locomotion in a wide variety of biological systems and environments.

6:47PM P07.00008 Hysteresis Behavior of Large Scale Structure in Turbulent Rotating Plane Couette Flow1, YUHAN HUANG, State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing 100871, China, ZHENHUA XIA, Department of Engineering Mechanics, Zhejiang University, Hangzhou 310027, China, MINPING WAN, SHIYI CHEN, Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, Shenzhen 518055, China — The existence of counter rotating vortex pairs is important in rotating plane Couette flow (RPCF). Two groups of numerical simulation were conducted with rotation number, \( Ro (Ro = 2\Omega b/U_w) \), with \( \Omega \) being the constant angular velocity in the spanwise direction, \( b \) being half channel height and \( U_w \) being half wall velocity difference, varying between 0.01 and 0.6. We found that the number of vortex pairs in a finite computation domain exhibits a hysteric behavior as \( Ro \) increases and decreases. When \( Ro \) decreases from 0.03 to 0.3, the number of vortex pairs is 2. When \( Ro \) decreases from 0.3 to 0.03, the number of vortex pairs is 3. This phenomenon is related to multiple states in RPCF or state bifurcation. Turbulent statistics such as friction, turbulent kinetic energy also form hysteresis loops. A linear stability analysis is performed on this problem to explain the formation of the hysteresis behavior.

1This work was supported by the National Science Foundation of China (Grants No. 11822208, No. 1177297, No. 11672123, and No. 91752201)

Monday, November 25, 2019 5:16PM - 7:13PM — Session P08 Non-Newtonian Flows II

5:16PM P08.00001 Entrance effects and high shear rate rheology of shear banding wormlike micelle fluids in microcapillary flow1 — PAUL SALIPANTE, VISHNU DHARMARAJ, STEVEN HUDSON, National Institute of Standards and Technology — The viscosity of a shear-banding wormlike micelle solution at high shear rates is investigated using capillary rheology and particle streak velocimetry. Measurements of the flow profile and pressure gradient show an extended entrance region, which exceeds a length to diameter ratio of 100, to reach a fully developed flow. We characterized this entrance region for capillaries with different cross-sections and used the results to select a downstream portion of the capillary where viscosity measurements can be made on fully developed flow. Measurements from this portion of the channel show a shear-thinning power-law behavior for all channel geometries from shear rates of 1,000 1/s to 120,000 1/s. Varying the surfactant concentration shows two distinct power-law behaviors that depend on both shear rate and concentration and are an indication of change in micelle length.

1NIST on a Chip

5:29PM P08.00002 Gels formed by worm-like micelles: Yielding and fluidization characteristics, RONAK GUPTA, RODRIGO MITISHITA, GWYNN ELFRING, Dept. of Mechanical Engineering, University of British Columbia, IAN FRIGAARD, Departments of Mathematics and Mechanical Engineering, University of British Columbia — A rarely studied system of long chain surfactant molecules has been shown to display ‘gel-like’ rheology at room temperature despite the absence of cross-linking. Using a similar model system, we investigate the yielding characteristics of micellar gels using creep and amplitude sweeps and elucidate dependencies on temperature and surfactant concentration. Further, we study the phenomenon of shear driven fluidization to better quantify the solid-liquid and yielding transitions in worm-like micellar gels.

5:42PM P08.00003 Linear and non-linear rheological study of long-chained surfactant solutions for gravel packing operations, RODRIGO MITISHITA, RONAK GUPTA, GWYN ELFRING, Department of Mechanical Engineering, University of British Columbia, IAN FRIGAARD, Departments of Mechanical Engineering and Mathematics, University of British Columbia — Long chained zwitterionic surfactants are extensively used in the oil industry, such as fracturing and gravel packing operations. Specifically in the context of gravel packing, the surfactant solutions need high viscosities over a wide range of shear rates and temperatures to successfully carry the gravel particles. The surfactant fluids are also drag-reducing due to their viscoelasticity, which aids in the transport over long distances at high pumping rates. In this study, we characterize a commercial surfactant for gravel packing by carrying out linear and non-linear rheology experiments. We show that the surfactant-water system exhibits a gel-sol transition with temperature. The results also quantify how the relaxation time of the solution and elastic modulus of the gel state depend on surfactant concentration and temperature. Finally, we discuss implications of rheology on gravel packing and turbulent drag reduction.

5:55PM P08.00004 Experimental method to measure liquid extensional properties during atomization, MARIE-CHARLOTTE RENOULT, CORIA-UMR 6614 INS A Rouen Normandy, CHRISTOPHE TIREL, CORIA-UMR 6614 Rouen University, CHRISTOPHE DEMOUCHET, CORIA-UMR 6614 CNRS — An experimental method has been developed to measure the extensional properties of a viscoelastic polymer solution experiencing atomization. It is based on the statistical and 3D multiscale analysis of the capillary thinning of liquid ligaments controlled either by elasticity or viscosity. The method principle will be first presented and validated using an ensemble of emulated ligaments. Then, the method procedure will be described from a free jet imaging setup to the extraction of the liquid extensional properties, i.e. the relaxation time and the terminal elongational viscosity. This procedure will be applied to a series of experiments conducted with several viscoelastic solutions, flow rates and quasi-cylindrical nozzles. Finally, the influence of the polymer concentration, of the jet velocity and of the nozzle dimensions on viscoelastic properties will be discussed.

6:08PM P08.00005 Taylor-Couette flow of shear-thinning and viscoelastic polymer solutions, NEIL CAGNEY, School of Engineering and Materials Science, Queen Mary University of London, Mile End Road, London, E1 4NS, UK, TOM LACASSAGNE, STAVROULA BALABANI, Department of Mechanical Engineering, University College London, Torrington Place, London, WC1E 6BT, UK — We study Taylor-Couette (TC) flow of a glycercol-water mixture containing xanthan gum (0 to 2000 ppm concentration). Both shear-thinning and viscoelasticity are induced, to assess the effect of the changes in rheology on various flow instabilities. The Reynolds number is slowly increased, and the flow is monitored continuously using flow visualisation [Cagney and Balaban, Phys. Fluids, 2019]. Shear-thinning is found to suppress many phenomena observed in previous studies of viscoelastic TC flow (e.g. diwhirls, disordered oscillations [Groisman and Steinberg, Phys. Rev, 1997]). The addition of polymers reduces the critical Reynolds number for the formation of Taylor vortices, but delays the onset of wavy flow. In the viscoelastic regime, the flow becomes unsteady soon after the formation of Taylor vortices, waviness changing with Reynolds number. Vortices are found to merge as Reynolds number increases, with the number of mergers increasing with polymer concentration. These changes in wavelength are hysteretic and occur in both steady and wavy regimes. Vortices in moderate and dense polymer solutions undergo a gradual drift in both their size and position, which appears to be closely linked to the splitting and merger of vortices.

6:21PM P08.00006 Co- and Counter-rotating Taylor-Couette Flows with Polyelectrolyte Solutions, VISHAL PANWAR, ATHENA METAXAS, CARI DUTCHER, University of Minnesota — Taylor-Couette flow, which is flow between two concentric, rotating cylinders, is ideal for studying flow behavior of complex solutions due to the wide variation of hydrodynamic flow states available. The addition of non-Newtonian polymer solutions increases the solution’s elasticity, which in turn can modify flow states that are typically dominated by inertial forces. In this study, a cationic polyacrylamide was used to modify the elasticity of the solution, and with varying concentrations of NaCl to alter the ionic strength of the solution. The coil conformation and relaxation times of charged polymers change depending on the ionic strength, from a more rigid conformation at low ionic strengths to a more flexible conformation at high ionic strengths, different in non-Newtonian responses to shear. The effect of polymer conformation as a result of varying solution ionic strength on TC flows with co- and counter-rotation of the cylinders will be discussed. In general, increasing ionic strength decreases elasticity, resulting in a shift towards a more Newtonian-like flow behavior. As the polymer conformation becomes more flexible, an overall trend of delayed transitions of flow states was observed.

6:34PM P08.00007 Viscosity measurement method of non-Newtonian fluids in pressure-driven flows based on energy dissipation rate, WOOK RYOL HWANG, HYE KYEONG JANG, Gyeongsang National University, SUN OK HONG, Ajou University, SANG BOK LEE, Korea Institute of Materials Science, JU MIN KIM, Ajou University — A novel viscosity measurement method is presented, which is established on the balance of the energy dissipation rate such that the external power is dissipated within the system as viscous dissipation in a laminar. The effective viscosity can be expressed algebraically in terms of the pressure drop and flow rate and the corresponding effective shear rate is readily determined by flow rate; the relationship between effective viscosity and effective shear rate is found identical to the true material viscosity behavior. The two flow numbers, which depend on flow geometry only and are almost independent of fluid rheology, are involved: the coefficient of energy dissipation rate that associates the total energy dissipation rate to the Reynolds number; and the coefficient of effective shear rate, which relates flow rate to effective shear rate. Three different flows with complicated geometries were tested: numerical validation for axisymmetric expansion-constriction flows and flows in a Kenics mixer, and experimental validation for flows in a complex microfluidic array with Xanthan gum solutions.

6:47PM P08.00008 Impacts of geometric parameters on bulge structure appearance in a cavity swept by a visco-elastic fluid, HIROSHI SUZUKI, MASAKI KAWATA, RURI HIDEMA, Department of Chemical Science and Engineering, Kobe University, FLUIDS AND PARTICLE ENGINEERING LABORATORY TEAM — We previously reported a bulge like flow structure is formed in a cavity swept by a visco-elastic fluid. In this study, effects of geometric parameters of the cavity on the bulge structure appearance have been investigated by flow visualization. As visco-elastic fluids, cationic surfactant aqueous solutions were treated with counter-ion suppliers of sodium salicylate. The molar ratios of sodium salicylate to cationic surfactants, oleylbishydroxyethylmethyammonium chloride, were changed from 1.5 to 10. The geometric parameters were also changed. The cavity depth, the wide flow path and the cavity length raged from 5 to 30 mm, from 25 to 40 mm and from 50 to 100 mm, respectively. From the results, three kinds of flow regimes were observed: 1. simple separation and reattachment occurrence, 2. bulge structure appearance and 3. recirculation region covering the cavity. In the cases when the molar ratio is high, bulge structure was not found to appear. When the aspect ratio of the narrow flow path was high, the flow also transited directly from 1 to 3 of the above flow regime. This flow regime was concluded to relate to the Deborah number of fluids.
resources from Consortium des Equipements de Calcul Intensif (CECI).

The position of the leader's wake is estimated and used as the target of the autopilot to maintain the follower in the optimal position. The efficiency is studied through the analysis of the related aerodynamic forces and the resulting wake. Finally, based on an Ensemble Kalman Filter, the robustness of the autopilot is verified with the LES of a single aircraft in turbulent flow. Then the influence of a leader's wake on a follower's dynamics is studied through the simulation of the aircraft dynamics and the LES of the wakes by means of a Vortex Particle-Mesh method. In a 2-ship formation, the distances between the vortex cores get altered in a random fashion. At $kh = 5$, the onset of jet switching in the trailing wake gets advanced $kh = 1$. The flow reattachment onsets inboard while the flow remains separated from the leading edge further outboard. Under certain axial acceleration magnitudes, a long whip-like nFTLE ridge lifting off the wing surface can be observed. The root of the whip-like structure moves toward the trailing edge and it closely follows a local chordwise pressure peak. It can be inferred that the nFTLE ridges’ roots across the span form a line that moves along with the high pressure zone in response to the axial gust. Following this region downstream along the chord is a region of low pressure, consistent with lift being re-established on the wing as the flow reattaches. Vertical accelerations will be studied and combined with axial acceleration cases to obtain further insight of flow structure evolution in highly unsteady environments for non-slender swept wings.

1This work was supported by the Office of Naval Research under ONR Award No. N00014-16-1-2732.

5:42PM P09.00003 Large Eddy Simulations of controlled aircraft in formation relying on wake sensing,¹, IGNACE RANSQUIN, DENIS-GABRIEL CAPRACE, PHILIPPE CHATELAIN, Universite Catholique de Louvain, UCLouvain, JEFFREY ELDREDGE, UCLA — Aircraft formation flight leads to substantial improvements in energetic efficiency even for large separations. Maintaining an energy-saving formation requires the estimation of the preceding aircraft’s wake position, and implies a sensing strategy. We propose to leverage the measurements of the six degrees-of-freedom dynamics of the follower aircraft. This study combines the simulation of the aircraft dynamics and the LES of the wakes by means of a Vortex Particle-Mesh method. In a 2-ship formation, the aerodynamics and vorticity sources are modeled using an immersed lifting line approach. The follower aircraft operates an autopilot in the form of a hierarchy of controllers that govern the ailerons, rudder, elevator and thrust in order to achieve wake sensing and tracking. The robustness of the autopilot is verified with the LES of a single aircraft in turbulent flow. Then the influence of a leader’s wake on a follower’s dynamics is studied through the analysis of the related aerodynamic forces and the resulting wake. Finally, based on an Ensemble Kalman Filter, the position of the leader’s wake is estimated and used as the target of the autopilot to maintain the follower in the optimal position. The efficiency of the tracking procedure is analyzed and shows promising results.

¹Funding from the European Research Council (ERC) under the European Unions Horizon 2020 research program; Computational resources from Consortium des Equipements de Calcul Intensif (CECI).
Timing compared to the starting and gust motions is shown to be important for the lift performance. Direct lift measurements are made via a transducer. The influence of varying the main-wing aspect ratio is also examined. The panel actuation promotes main-wing leading-edge vortex shedding, which should decrease the lift as desired. To test the effect of the modified flow on the force, towing-tank experiments to study the unsteady lift force produced by a high-angle-of-attack translating wing with a tip panel that rotates inward in the wing plane. The goal is to examine whether combined outboard sweep and rotation can influence the lift for two types of configurations such as different VG's angle of attack with respect to the air flow and different slot topologies were compared and discussed.

Interaction of the Leading Edge Vortex and Shear Layer Vortices for an Airfoil Undergoing Dynamic Stall

The flow field around a NACA0012 airfoil undergoing large amplitude sinusoidal pitching is investigated using Particle Image Velocimetry (PIV). The airfoil is pitched symmetrically about the quarter chord point with a pitch angle of 40, 30, and 20 at a reduced frequencies of =0.2-0.6 and Re =12000. In all cases a strong leading edge vortex (LEV) is formed with 2-3 weaker shear layer vortices (SLV) also forming along the airfoil surface. In some cases the LEV and the nearest SLV combine. In other cases SLV's combine. And finally in some cases the vortices remain independent of each other. The data suggest the formation a topological saddle forms between the two vortices that correlates with events in which the two vortices combine. The saddle appears below the center of the weaker vortex and moves relative to the vortices as they interact. The vortices are found to combine when the saddle moves above the center of the weaker vortex. The vortices remain distinct when no saddle appears. The data show that the interaction of the LEV and its nearest SLV occurs only for a limited range of the parameter space investigated.

6:34PM P09.00007 Wake transitions of flexible foils in a viscous uniform flow

We perform numerical simulations for two types of rigid and flexible thin foils in a viscous uniform flow to explore the effect of flexibility on wake structures. The thin foils are prescribed by the heaving oscillation motions and the relevant non-dimensional parameters are the chord length based Strouhal number and flapping amplitude. When the dynamical features of the flow wakes are varied with respect to the two parameters, it is possible to make a direct comparison between a rigid and flexible thin foils. The wake transition boundaries of the rigid thin foil are predicted by constant amplitude based Strouhal number lines, consistent with previous studies. However, contrary to the observation from the rigid thin foil, the wake transition boundaries of the flexible thin foil are not predictable by constant amplitude based Strouhal number lines. We find that the sum of the leading and trailing edge circulations plays an important role to determine a wake pattern behind a rigid and flexible foil, and wake transitions are observed beyond critical circumulations. 

6:47PM P09.00008 The Lift of a Translating Plate with Tip Sweep and Rotation

The lift of a translating plate with tip sweep and rotation are examined. This work could provide a method for drones to temporarily boost lift or outward in the wing plane. The goal is to examine whether combined outboard sweep and rotation can influence the lift for two types of motions: a starting flow and positive/negative 1/2-sine streamwise gusts. This work could provide a method for drones to temporarily boost lift during high-angle-of-attack maneuvers and reduce lift perturbations from streamwise gusts. For the starting flow, our prior flow visualization indicates that early outward panel motion produces a new swept-edge vortex (SEV) and stretches the tip vortex (TV) and trailing-edge vortex outboard, which should enhance lift. For a gust-like forward surge, inward panel motion prior to the gust sheds the panel SEV and TV, and promotes main-wing leading-edge vortex shedding, which should decrease the lift as desired. To test the effect of the modified flow on the force, direct lift measurements are made via a transducer. The influence of varying the main-wing aspect ratio is also examined. The panel actuation timing compared to the starting and gust motions is shown to be important for the lift performance.

This work is supported by the National Science Foundation, award no. CBET-1706453, supervised by Dr. Ronald Joslin.

Monday, November 25, 2019 5:16PM - 7:13PM – Session P10 Nonlinear Dynamics: Model Reduction III

3A - Benjamin Herrmann, University of Washington

This work is supported by the National Science Foundation, award no. CBET-1706453, supervised by Dr. Ronald Joslin.
5:16PM P10.00001 Time-frequency analysis of intermittent coherent structures in turbulent flows, AARON TOWNE, PEIJING LIU, University of Michigan, FLOW MODELING AND CONTROL TEAM — Time-frequency analysis is a popular and powerful tool for identifying and analyzing intermittent events in fluid systems. Typically, these analyses are carried out by computing spectrograms or scalograms from the time history of a single flow variable at a single location in space, e.g., one component of velocity at some location of interest. A disadvantage of this approach is that the relationship between spectrograms or scalograms computed at different points in the flow or for different flow quantities is ambiguous, and, in particular, the role of coherent structures in producing the observed intermittency is obscure. In this presentation, we introduce an approach for generating global spectrograms and scalograms that describe the time-frequency behavior of coherent structures defined over all flow variables and spatial locations. The method is based on two extensions of spectral proper orthogonal decomposition, which defines flow structures that evolve coherently in space and time (Towne et al., J. Fluid Mech. Vol. 847, 2018). We demonstrate the method using the example of coherent wavepacket structures in a turbulent jet.

5:29PM P10.00002 A modal decomposition for discovery of nonlinear triadic interactions from flow data, OLIVER SCHMIDT, University of California San Diego — We seek a decomposition of a flow field into spatial modes that reveal the footprint of triadic interactions. This goal is accomplished by extending bispectral signal analysis to multidimensional datasets with at least one inhomogeneous spatial direction. As we are interested in finding the dominant large-scale coherent structures associated with quadratic nonlinearities, we require the modal decomposition to optimally represent the data in terms of its phase coherence, as characterized by the third-order cumulant. In the frequency domain, this higher-order statistic is readily computed from a product of Fourier transforms at different frequencies. An algorithm for the computation of the proposed bispectral mode analysis is presented and applied to three data sets obtained from direct numerical simulation of cylinder flow at Re=300, large-eddy simulation of transitional jet flow, and particle image velocimetry measurements of a massively separated flat plate at a high angle of attack. The results are visualized in terms of frequency-frequency plots similar to the bispectrum that indicate the presence of the quadratic phase coupling, and spatial modes that can reveal the nature of the nonlinear interaction.

5:42PM P10.00003 Real-Time Reduced Order Modeling Of Deterministic and Stochastic Systems Using Time-Dependent Low-Rank Basis, HESSAM BABAE, MICHAEL DONELLO, University of Pittsburgh — We present a scalable method for the extraction of a time-dependent basis from observations of deterministic/stochastic systems. This method is based on a variational principle whose optimality condition leads to a closed-form evolution equation of the basis. The method is scalable with respect to the size of the data and the reduction order. We will also present real-time reduced order model, which are constructed from projections of the full-dimensional dynamics onto the time-dependent basis. We will present two case studies for the reduced-order modeling of: (1) transient instabilities in Kuramoto-Sivashinsky equation, and (2) transient flow over a bump.

5:55PM P10.00004 Data-driven modeling of chaotic flows with non-Gaussian statistics, HASSAN ARBABI, THEMISTOKLIS SAPSIS, Massachusetts Institute of Technology — We present a data-driven framework for modeling chaotic flows in the form of linear stochastic differential equations (SDEs) observed under a nonlinear map. We discover the nonlinear map using the theory of optimal transport for measures, and identify the system of SDEs by matching the spectral density of the optimally transported data. The inclusion of the nonlinear observation map allows us to build models of chaotic systems with moderate dimensions (e.g. 10 and more) that capture non-Gaussian features of the invariant measure including skewness and heavy tails. We demonstrate the application of our framework through a few examples including a high-Reynolds cavity flow and observational climate data.

6:08PM P10.00005 Reduced order flow control using input-output hidden Markov model, PALASH SASHITTAL, DANIEL BODONY, University Of Illinois at Urbana-Champaign — In this work we extend the cluster based reduced order modeling framework to partially observed fluid systems with a control oriented perspective. We employ a data-driven model learning approach coupled with a closed-loop control strategy. Model parameters are learned using expectation maximization in the presence of scarce and noisy data. Two feedback controller design methods are proposed to control the long term behavior of the system. We demonstrate the performance of the controllers by controlling the transitions of the Lorenz system and to suppress the nonlinear vortex shedding past an inclined flat plate. The application of hidden Markov model based control to primary atomization of a liquid jet in a high-speed gas co-flow will be discussed.

1This work was sponsored by the Office of Naval Research (ONR) as part of the Multidisciplinary University Research Initiatives (MURI) Program, under grant number N00014-16-1-2617.

6:21PM P10.00006 Autoencoders for discovering coordinates and dynamics from data, KATHLEEN CHAMPION, University of Washington, BETHANY LUSCH, Argonne National Laboratory, NATHAN KUTZ, STEVEN BRUNTON, University of Washington — Understanding and efficiently modeling high-dimensional scientific data such as fluid flows require methods for identifying interpretable reduced models from the data. Deep learning methods are extremely flexible, showing a remarkable ability to behave as universal function approximators and demonstrating impressive performance at modeling and predictive tasks; however the resulting models have many parameters and are not constrained. A new family of linear model discovery (SINDy) provide parsimonious interpretable models, but require knowledge of the proper coordinates for representing dynamics. In this work, we present a method that combines an autoencoder network with SINDy for the simultaneous discovery of reduced coordinates and parsimonious dynamical models from high-dimensional data. The resulting models are interpretable, containing a small number of active terms that describe the dynamics in the reduced space. We demonstrate the success of this approach on data from a number of example systems.

6:34PM P10.00007 Identifying coarse-grained PDE models from data with the sparse identification of nonlinear dynamics, KADIERDAN KAHEMAN, EURIKA KAISER, ADITYA NAIR, NATHAN KUTZ, STEVEN BRUNTON, University of Washington — Identifying reduced order models of complex systems, such as are found in fluid mechanics, is a challenging, yet vital topic. In addition to Galerkin projection, which has been a mainstay of the fluid mechanics' community for decades, techniques from machine learning are emerging to identify accurate and efficient models from data. The sparse identification of nonlinear dynamics (SINDy) algorithm has recently been shown to provide data-driven reduced-order models for fluids that are both interpretable and generalizable. However, it is difficult to identify models with rational function nonlinearities using the SINDy approach, although these dynamics appear in many systems of interest, especially those systems with a separation of timescales. A recent extension to SINDy has enabled the identification of rational function nonlinearities, but this approach is numerically fragile and highly sensitive to noise. In this work, we develop a robust extension of SINDy to identify rational function nonlinearities, using an iterative approach to improve the conditioning of the algorithm. We demonstrate this implicit SINDy approach on several ODE and PDE systems, including the unsteady BZ chemical flow reaction.
6:47PM P10.00008 A study of state-of-the-art model reduction techniques applied to flow simulations with moving immersed boundaries. SERENA COSTANZO, TARANEH SAYADI, Sorbonne Université, PASCAL FREY, Sorbonne Université — Reducing a detailed system into smaller sized models, capable of reproducing the main features and dynamics of the original configuration is a common practice in optimisation and control community, which could also serve as a way to make function evaluations less expensive. Since control is one of the future applications of this study it is necessary to identify the most suitable methodology applicable to detailed Navier-Stokes simulations for prediction purposes. To this end various strategies such as the GNAT method (Gauss-Newton with approximate tensors), POD with Galerkin regression method, implemented with the machine learning algorithm SINDY (Sparse Identification of Nonlinear Dynamics), and conventional POD-DEIM methods are compared in the context of an incompressible Navier-Stokes solver with moving immersed boundaries capabilities. The capability of these methods to interpolate between various operating conditions and to extrapolate the solution is investigated. The ability of each reduction strategy in dealing with existing nonlinearities and moving immersed boundaries is also identified.

7:00PM P10.00009 Learning interpretable stochastic models with sparse regression1. JARED CALLAHAM, University of Washington, JEAN-CHRISTOPHE LOISEAU, Arts et Metiers ParisTech, J. NATHAN KÜTZ, STEVEN BINTUM, University of Washington — Learning interpretable stochastic models from high-dimensional data is a relevant task in many fields such as climate science, genomics, or turbulence. Recent advances in machine learning such as approximate inference from stochastic differential equations (SDEs) have demonstrated the effectiveness of neural networks to model complex systems. However, the over-parametrization of these models renders them uninterpretable from a physical point of view. In this work, we develop a methodology that allows to automatically discover interpretable stochastic models by learning reduced and interpretable SDEs directly from data. We describe an approach to learning interpretable stochastic models, devoting particular attention to practical considerations such as low sampling rates and time-correlated forcing. We apply our approach to prototypical nonlinear dynamics and demonstrate more accurate recovery than existing stochastic model discovery methods.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P11 Experimental Techniques: Quantitative Flow Visualization I 3B - Sam Grauer, Georgia Tech

5:16PM P11.00001 Three dimensional velocity and pressure measurements in a turbulent shear layer behind a backward-facing step.2. KARUNA AGARWAL, OMRI RAM, JIN WANG, YUHUI LU, JOSEPH KATZ, Johns Hopkins University — This study characterizes the unsteady pressure field generated by quasi-streamwise vortices that develop between the main spanwise vortices in the near field of a shear layer behind a backward facing step. Our objective is to understand the preferential location of pressure minima moves from the spanwise vortices at low Re to the core of quasi-streamwise vortices located between 45% and 75% of the reattachment length forming 1-2 mm wide and 5-7 mm long cavities. Tomographic imaging followed by 3D particle tracking using the Shake-the-Box method is used for calculating the instantaneous velocity and acceleration fields. These results are interpolated using singular value decomposition, and then refined using Constrained Cost Minimization to generate divergence free velocity and curl-free material acceleration at a spatial resolution of 250 m. The pressure is obtained by spatially integrating the material acceleration. The measurements are performed at Reynolds numbers based on separating boundary layer height Re=7100 and 17700. Analysis examines the effects of Re on the frequency, time evolution, size, strength and the pressure in the quasi streamwise vortices. The preferential location of pressure minima moves from the spanwise vortices at low Re to the quasi-streamwise structures at the higher Re.

1Funded by ONR

5:29PM P11.00002 3D Particle Reconstruction of Volumetric Particle Image Velocimetry with Convolutional Neural Network3. SHAOWU PAN, Department of Aerospace Engineering, University of Michigan, Ann Arbor, USA, QI GAO, School of Aeronautics and Astronautics, Zhejiang University, China, QIJIE LI, HONGPING WANG, Key Laboratory of Fluid Mechanics, Beihang University, China — 3D particle reconstruction of volumetric Particle Image Velocimetry (PIV) is an under-determined inverse problem of which exact solution is difficult to obtain. Traditionally, approximated solutions can be obtained via optimization, e.g., multiplicative algebraic reconstruction technique (MART). Despite its popularity in recent years, the performance of MART-like algorithms deteriorates when the particle concentration becomes high, e.g., particle per pixel (ppp) ≈ 0.3. In this work, a particle reconstruction method based on convolutional neural network (CNN) is proposed. The method consists of two steps: first, an initial particle field is generated from a plurality of two-dimensional particle images get by camera using multiplied line-of-sight (MLOS) method; second, we use CNN to take the aforementioned initial particle field as input and output the 3D reconstructed particle fields. The data is artificially generated by randomly placing particle in 3D space then projected to four cameras to obtain 2D particle images. Compared to the traditional MART algorithm, the proposed method is not only significantly improving the accuracy of 3D particle reconstruction especially at high particle concentration but also eight times faster than the traditional MART algorithm.

1AFOSR, ARO, NDSEG fellowship
2This work was supported by the NNSFC (grant No. 11472030, 11327202, 11490552) and FRFCU (Grant No. YWF-16-JCTD-A-05).
A High-Spatial-Resolution Three-Dimensional Three-Component Velocimetry Method Based on Divergence-Free Polynomials

Present a physics-constrained method of inferring three-dimensional three-component (3D3C) velocity fields in constant-density flows from reconstructed 3D Mie-scattering fields of tracer particles. The proposed method is based on the representation of the estimated velocity field as a linear combination of divergence-free polynomial basis functions; the piecewise constant representation of the estimated velocity field that is inherent to tomographic particle image velocimetry (T-PIV) is replaced by a smooth representation that automatically satisfies conservation of mass. The appropriate linear combination is determined using a non-regularized motion estimation framework that is influenced by optical flow estimation. We provide a detailed evaluation of the proposed method in terms of accuracy and spatial resolution, treating 3D constant-density DNS data as the ground truth. We show that the proposed method (a) yields significant improvements in accuracy and spatial resolution compared to an idealized implementation of T-PIV and (b) achieves comparable accuracy and spatial resolution to an idealized estimate that corresponds to the theoretical one-vector-per-particle limit.

This work was supported by the US Air Force Office Scientific Research under Grant FA9550-17-0011 (Project Monitor Dr. Chiping Li) and the National Science and Engineering Research Council of Canada through an Alexander Graham Bell Canada Graduate Scholarship.

Direct background-oriented schlieren tomography

We present a novel approach to background-oriented schlieren (BOS) tomography that combines the deflection sensing and reconstruction algorithms. BOS imaging is a refraction-based flow visualization technique. Simultaneous BOS measurements from multiple cameras can be reconstructed by computed tomography to estimate the fluid’s 3D refractive index field, which is post-processed to obtain local densities. Each camera is focused on a textured background pattern that is positioned behind the fluid. Density gradients cause distortions in the image; the deflected light trajectories are typically determined using an optical flow algorithm. These deflections constitute the projection data for reconstruction. Deflection sensing is itself a complex inverse problem and a primary source of error in BOS tomography. We propose an alternative measurement model for BOS tomography that incorporates the optical flow equation. The deflection model is extended to calculate image gradients, directly, such that the refractive index field is reconstructed from the distorted images. As a result, reconstructions must satisfy observed gradients instead of inferred deflections, which are prone to error. The talk describes our measurement model and presents a numerical assessment of direct BOS tomography.

Volumetric Velocimetry in the Rotating Frame of Reference using a Plenoptic Camera

A single plenoptic camera mounted coaxially above a hub mounted mirror that rotates with the rotor enables instantaneous and three-dimensional flow field measurements over a wide range of azimuth angles. The design and implementation of this concept as an experimental test facility to conduct flow field and force measurements over a rotating wing is discussed. In the presentation, an overview of the methodology along with the challenges, details of the test facilities and some preliminary flow field and force measurement results will be discussed.

Application of Plenoptic Camera 3D PIV in a Rotating Frame of Reference

A major limitation to the thorough understanding of a rotating flow’s aerodynamic characteristics is the difficulty in measuring data within the rotating frame of reference. Rotating 3D Velocimetry (R3DV) is a concept designed to acquire volumetric flow field measurements over a rotor, within the rotating frame of reference, using a single camera. A submerged wing rotates in conjunction with a mirror that reflects light traveling through flow over the wing to a stationary plenoptic camera. The plenoptic camera’s added array of microlenses between the aperture and image sensor gives it the ability to capture the incident angle of light rays, allowing for 3D reconstruction of volumes. Traditionally, plenoptic camera calibration has been explored for a stationary field of view. In current development is a method to incorporate rotation into the calibration such that images from all azimuth angles can be calibrated by interpolation, thereby eliminating the need to calibrate each distinct angle. Other challenges include achieving sufficient depth resolution and obtaining an adequate depth of field at a distant focal plane. Preliminary results demonstrating the efficacy of the R3DV method will be presented along with explanations of how the aforementioned problems will be overcome.

The authors would like to acknowledge support from the Army Research Office under the Grant Number: W911NF-19-1-0052.

Technique for Characterization of 3D Unsteady Fluid-Structure Interactions via a Single Plenoptic Camera

Auburn University — Due to flexible boundaries, many flows are unsteady, three-dimensional and subjected to fluid-structure interactions (FSI). Conventional studies of FSI nominally treat the flow and boundary surface measurements separately, resulting in a loss of true FSI physics, especially for aperiodic flows. More recently, simultaneous multi-camera approaches using tomographic-PIV/3D-PTV and Digital Image Correlation (DIC) have been implemented. This approach in this paper is to use a single plenoptic camera equipped with a microlens array, plenoptic cameras capture a volume’s light-field in 4D, which enables single-camera 3D measurements. In this study, a newly developed kHz-rate plenoptic camera is applied to characterize the FSI of a flexible 50x 30 x 30mm$^3$ tube model. Macroscopic painted dots were used to track surface motion, while micron-sized particles were used as the flow tracer. A pulsatile flow was applied to intermittently collapse the tube, during which the dots and seeds were simultaneously imaged. In post-processing, dots and seeds are separated by diameters and/or shape, after which their 3D trajectories are tracked separately via a plenoptic-PTV algorithm. Preliminary proof-of-concept results are presented, along with further analyses to optimize particles/dots sizes and densities.

The authors would like to acknowledge support from the Army Research Office under the Grant Number W911NF-19-1-0052.

6:47PM P11.00008 Technique for Characterization of 3D Unsteady Fluid-Structure Interactions via a Single Plenoptic Camera, BRIAN THUROW, ZU TAN, BIPIN TIWARI, VRISHANK RAGHAV, Department of Aerospace Engineering, Auburn University — Due to flexible boundaries, many flows are unsteady, three-dimensional and subjected to fluid-structure interactions (FSI). Conventional studies of FSI nominally treat the flow and boundary surface measurements separately, resulting in a loss of true FSI physics, especially for aperiodic flows. More recently, simultaneous multi-camera approaches using tomographic-PIV/3D-PTV and Digital Image Correlation (DIC) have been implemented. This approach in this paper is to use a single plenoptic camera equipped with a microlens array, plenoptic cameras capture a volume’s light-field in 4D, which enables single-camera 3D measurements. In this study, a newly developed kHz-rate plenoptic camera is applied to characterize the FSI of a flexible 50x 30 x 30mm$^3$ tube model. Macroscopic painted dots were used to track surface motion, while micron-sized particles were used as the flow tracer. A pulsatile flow was applied to intermittently collapse the tube, during which the dots and seeds were simultaneously imaged. In post-processing, dots and seeds are separated by diameters and/or shape, after which their 3D trajectories are tracked separately via a plenoptic-PTV algorithm. Preliminary proof-of-concept results are presented, along with further analyses to optimize particles/dots sizes and densities.
orifice-generated vortex rings

Trailing vortices merge with the primary ring to grow into a well-formed ring at an optimal formation time. Energy consideration based on the total circulation compared to a nozzle. In the case of orifice-generated vortex rings, the formation number has been found to range between 1.0 to 2.0. Furthermore, the radial component of velocity drastically destabilises the flow which results in an earlier pinch-off of the primary ring.

With the experimental data.

gases after bullet ejection is overlaid with the predicted flow field which reveals satisfactory agreement. Gunshot residue is an important factor and can either propel blood drops further from the target, or even turn them backwards toward the target. An image of the propagating muzzle backward blood spatter resulting from a gunshot.

Turbulent vortex rings of propellant gases skew the distribution of blood stains on the ground are an important phenomenon to consider in crime scene reconstruction. It is shown that this has a significant repercussion on the outcome of passive admixture by turbulent vortex rings. Turbulent vortex rings of propellant gases originating from the muzzle of a gun after a gunshot are a moment/torque on the vortex and the corresponding conservation is ensured by the appropriate vorticity flux into the core and throughout the sheet. The effect of this constraint on the total circulation in the sheet and the location of the vortex are presented for the roll-up of self-similar vortex sheets.

5:29PM P12.00002 Three-dimensional Vortical Structures in a Curved Pipe under Fully Developed Pulsatile Inflow

We numerically investigate spatial and temporal evolution of multiple three-dimensional vortices under a fully developed cardiac physiological (pulsatile) inflow of a Newtonian fluid in a 180 degree rigid pipe with circular cross-section, without taper or torsion. We identify vortical structures using vortex identification methods and characterize their evolution throughout the deceleration phase, capturing both Dean- and Lyne-type vortices for which the planes of rotation are different. We track trajectories of Dean-type vortices and find agreement with experimental results. We also show the connection along the axial direction between regions of organized vorticity observed at various cross-sections, which previous 2-D analysis could not provide, and demonstrate that the combined effect of geometry curvature and deceleration produces a strong helical flow that is conducive to the formation of a single pair of Dean-type counter-rotating vortices that is connected to the classical Dean vortex pair near the entrance to the curve. Understanding the formation of these vortical structures is necessary to draw any correlation between the flow and wall shear stress distributions produced under physiological conditions.


Alexander Yarin, University of Illinois Chicago — Self-similar turbulent vortex rings are investigated theoretically in the framework of the semi-empirical turbulence theory for the modified Helmholtz equation. The velocity and vorticity fields are established, as well as the transport of passive admixture by turbulent vortex rings. Turbulent vortex rings of propellant gases originating from the muzzle of a gun after a gunshot are an important phenomenon to consider in crime scene reconstruction. It is shown that this has a significant repercussion on the outcome of backward blood spatter resulting from a gunshot. Turbulent vortex rings of propellant gases skew the distribution of blood stains on the ground and can either propel blood drops further from the target, or even turn them backwards toward the target. An image of the propagating muzzle gases after bullet ejection is overlaid with the predicted flow field which reveals satisfactory agreement. Gunshot residue is an important factor in determining the events of a violent crime due to a gunshot and are considered to be entrained and transported by the propellant gases. The self-similar solutions for the flow, vorticity, and concentration of gunpowder particles are predicted and the results are shown to be consistent with the experimental data.

5:55PM P12.00004 Formation number, pinch-off time and optimal formation time of orifice-generated vortex rings

Raphael Limbour, Jovan Nedic, McGill University — Vortex rings are generated experimentally by impulsively discharging fluid through a sharp-edge nozzle or orifice. As the actuator pushes the flow through the tube, the shear layer at the outlet rolls-up, detaches and propagates downstream in one or several vortices from which we can define the pinch-off time, the formation number and an optimal formation time. The radial component of velocity at the exhaust of an orifice results in a significant increase in the final circulation, the kinetic energy and the hydrodynamic impulse (Krieg & Mohseni, 2013). Measurements show a 140% increase in the total circulation compared to a nozzle. In the case of orifice-generated vortex rings, the formation number has been found to range between 1.0 to 2.0. Furthermore, the radial component of velocity drastically destabilises the flow which results in an earlier pinch-off of the primary ring. Trailing vortices merge with the primary ring to grow into a well-formed ring at an optimal formation time. Energy consideration based on the Frenkel-Norbury family of vortex rings accurately predicts the formation number for orifice-generated vortex rings.
6:08PM P12.00005 Accurate and efficient coupling of a near-wall Eulerian solver with a Vortex Particle-Mesh method for aerodynamics and wakes\textsuperscript{1}, PHILIPPE BILLUARD, PHILIPPE CHATELAIN, GREGOIRE WINCKELMANS, Universite catholique de Louvain — We present and illustrate a new hybrid Eulerian-Lagrangian approach for aerodynamics: the near-wall region is resolved using an Eulerian solver (here finite differences) while the vortex-particle-mesh (VPM) method, supplemented by an immersed interface method (IIM), is used for capturing the wake region. Indeed, the isotropic elements of the VPM-IIM do not permit to resolve accurately and efficiently the boundary layer but perform very well in the wake region thanks to their negligible dispersion and diffusion. The grid-based solver is well suited for resolving the boundary layers. With such coupling, the advantages of both solvers are kept whereas their respective drawbacks are eliminated, permitting to simulate efficiently high Reynolds number flows. The key feature of the approach also lies in the way both methods are coupled: accurately and without Schwarz iteration. The approach is tested and analysed/validated on the flow past a cylinder. It is then applied to the flow past a regularized Joukowski airfoil.

\textsuperscript{1}The development work benefited from the computational resources provided by the supercomputing facilities of the Universite catholique de Louvain (CISM/UCL) and the Consortium des Equipements de Calcul Intensif en Federation Wallonie Bruxelles (CECI) funded by the Fond de la Recherche Scientifique de Belgique (F.R.S.-FNRS) under Convention No. 2.5020.11.

6:21PM P12.00006 Investigating the Van der Pol analogy in vortex shedding using Pade approximants and compact finite differences, DANIEL JOHNSTON, MOHAMMED AFSAAR, ADRIAN SESCU, IOANNIS KOKKINAKIS, University of Strathclyde — The Van der Pol (VDP) oscillator serves as an analogy for vortex shedding in the near-wake region of doubly infinite (i.e. spanwise homogeneous) slender bluff bodies. In this work, approximate solutions to the VDP equation are investigated using Pade approximants and compact finite difference schemes. The Pade approximant formulae are found using expansions of the well-known asymptotic solution. We compare this to the corresponding Taylor series and the classical 4th order Runge-Kutta solution. The Pade approximant solutions consistently show closer agreement to the numerical solution over their corresponding Taylor series, especially at $O(1)$ values of the small parameter in the VDP equation, for a longer interval in time. We then assess the accuracy of the numerical solution using compact difference schemes. We find that the spectral-like schemes introduced by Lele (J. Comp. Phys. 103, p.16, 1992) show closer agreement with the VDP analogy, when compared with conventional schemes, as the initial rate of growth of the oscillations in the near-wake is increased. Finally, we consider this analogy as a model of the Von-Karman vortex street in the wake behind a cylinder, validate the results using CFD simulations, and discuss conditions for which its applicability may break down.

6:34PM P12.00007 Numerical investigation of the model on vortex reconnection, YOSHIHUMI KIMURA, Nagoya University, KEITH MOFFATT, University of Cambridge — Recently we have developed an analytical model for the finite singularity problem for the Naiver-Stokes equations \cite{930967,2}. In this model, two circular vortex rings of circulation $\pm \Gamma$ and radius $R = 1/\kappa$ are symmetrically placed on two planes inclined to the plane $x = 0$ at angles $\pm \alpha$. Under an assumption that the vortex Reynolds number, $R_{\kappa} = \Gamma/\nu$, is very large, we have derived a nonlinear dynamical system for the local behavior near the points of closest approach of the vortices. Careful numerical investigation of the dynamical system reveals that the magnitude of vorticity could take any large value for small viscosity but remains finite since the minimum core radius never becomes zero. The assumptions for this analysis are far beyond the ones that the current DNS could attain, but we are curious whether and how much DNS can verify the tendency of the analysis. We are going to show some preliminary results of DNS. \cite{930967} Towards a finite-time singularity of the Naiver-Stokes equations Part 1. Derivation and analysis of dynamical system, H.K.M. & Y.K. JFM (2019) 861, 930967. \cite{2} Towards a finite-time singularity of the Naiver-Stokes equations Part 2. Vortex reconnection and singularity evasion, H.K.M. & Y.K. JFM (2019) 870, R1.

6:47PM P12.00008 Interaction of Tubular Vortices as a Function of Contact Angle, OSCAR VELASCO FUENTES, Departamento de Oceanografia Fisica, CICESE, Mexico — We study the evolution of two equal tubular vortices, which initially touch each other, as a function of the angle formed by their centerlines at the point of contact. To this end we solve the vorticity equation in a triple-periodic domain with a vortex-in-cell method, using as initial conditions two helical vortices of equal circulation $\Gamma$, pitch $L$, radius $R$ and core radius $a$. The axes of the helices are parallel lines separated by a distance $2R + 2a$, so that the vortices touch each other at a single point within the numerical domain. At this point the vortices centerlines make an angle $\alpha = 180^\circ - 2 \tan^{-1}(L/2\pi R)$. We analyzed the flow evolution by monitoring the position and topology of iso-surfaces of vorticity magnitude as well as the distribution of fluid particles initially located within the vortices. Both methods yield the same results in the identification of the following regimes: for small angles ($\alpha \to 0^\circ$) the vortices merge in a time that increases with the angle, for large angles ($\alpha \to 90^\circ$) the vortices reconnect in a time that decreases with the angle, for intermediate angles the vortices exchange mass but keep their identity during the whole simulation.

7:00PM P12.00009 Point vortices with dynamic circulation\textsuperscript{1}, PHILIP J. MORRISON, The University of Texas at Austin — The intimate connection between scalar vorticity dynamics and point vortex dynamics is well known. Because of Kirchhoff, Kida, and many others, Hamiltonian point vortex dynamics has been widely studied in a variety of contexts. In this talk I will discuss a generalization: a new conservative system of point vortex dynamics that allows for dynamic circulations.

\textsuperscript{1}Supported by U.S. Dept. of Energy Contract # DE-FG05-80ET-53088.

Monday, November 25, 2019 5:16PM - 7:00PM – Session P13 Convection and Buoyancy-driven Flows: Simulation 304 - Roberto Verzicco, Twente
5:16PM P13.00001 Application of new unstructured geometry capability within FDS to simulation of outdoor smoke transport and deposition, MARCOS VANELLA, Fire Research Division, National Institute of Standards and Technology, Gaithersburg, MD, USA, ORIOL RIUS, HSE Department, CERN, 1211 Geneva 23, Switzerland, GLENN FORNEY, Fire Research Division, National Institute of Standards and Technology, Gaithersburg, MD, USA, EMANUELE GISSI, Corpo Nazionale dei Vigili del Fuoco, Italy, JASON FLOYD, Jensen Hughes, Rockville, MD, USA, SAVERIO LA MENDOLA, HSE Department, CERN, 1211 Geneva 23, Switzerland, RANDALL MCDERMOTT, Fire Research Division, National Institute of Standards and Technology, Gaithersburg, MD, USA — Over the years, the Fire Dynamics Simulator (FDS) has become one of the industry preferred tools for simulation of fire scenarios in design of indoor fire protection systems, forensic studies and wildland fires, among others. The flow solver of FDS evolves the Low Mach approximation equations for thermally driven buoyant flows by means of Large Eddy Simulation (LES), and standard discretization on block structured meshes. Recently, we have been developing the capability of FDS to simulate such flows over complex unstructured geometries via a cut-cell scheme and immersed boundary method. In this talk we will focus on the numerical scheme, and ongoing work on simulation of an outdoor smoke dispersion problem considering agglomeration and deposition in the kilometer scale CERN Meyrin campus. Details of the software implementation, simulation setup, and species transport and deposition results for a specific contaminant release scenario will be discussed.

5:29PM P13.00002 Effects of Equations of State Selection in Numerical Simulations of Supercritical Carbon Dioxide1, ELIZABETH RASMUSSEN, Mechanical Engineering Department, University of Washington, MICHAEL MARTIN, SHASHANK YELLAPANTULA, National Renewable Energy Laboratory, JOHN KRAMLICH, Mechanical Engineering Department, University of Washington — Supercritical carbon dioxide (scCO₂) is employed in a growing range of applications including novel material synthesis and advanced energy systems. However, a lack of understanding of how the complex behavior of scCO₂ near the critical point 304.25 K and 7.39 MPa affects the flow field, accompanied by numeric challenges of simulating under these conditions, limits the use of simulation as a predictive tool in these systems. Initial simulations using the high-fidelity Span-Wagner equation of state at a pressure of 8 MPa, fluid and wall temperatures ranging from 305 K to 390 K, and Reynolds numbers ranging from 0.1 to 35, show complex changes in flow and heat transfer. When simulations are repeated using the ideal gas law, Soave-Redlich-Kwong, and Peng-Robinson equations of state to the system, many of these effects are not captured. We compare the drastically different flow characteristics between non-ideal and ideal models as well as present on the computational cost of the varying degrees of accuracy.

1This work was facilitated through the use of advanced computational, storage, and networking infrastructure provided by the Hyak supercomputer system at the University of Washington. Funding also provided by the Exascale Computing Project (17-SC-20-SC), a collaborative effort of two U.S. Department of Energy (DOE) organizations, the Office of Science and the National Nuclear Security Administration. The research was performed using computational resources sponsored by the Department of Energy’s Office of Energy Efficiency and Renewable Energy and located at the National Renewable Energy Laboratory.

5:42PM P13.00003 Boussinesq approximation in rapidly rotating flows1, JAGMOHAN SINGH, H. M. BLACKBURN, Monash University, Australia, J. M. LOPEZ, Arizona State University, USA, A. J. SMITS, Princeton University, USA — Rotating thermal convection (RTC) comprises of a thermal buoyant plume surrounded by a swirling flow. RTC is fundamental to many geophysical and engineering flows including tornados, firewhirls, dust devils and gas turbine combustors. Over the decades, RTC has been studied experimentally and numerically and it is shown that the axial buoyancy due to gravity in the presence of swirling creates a large updraft and aids to the formation of these columnar flows. In numerical simulations, the gravitational buoyancy is commonly modelled via the Boussinesq approximation which ignores the density variations in the momentum equation except in the terms multiplied by the gravity. However, a similar approach leads to additional buoyancy terms due to centrifugal forces, Coriolis forces and the inertia. The buoyancy due to centrifugal forces has recently started gaining attention but the other buoyancy terms are still overlooked and ignored. In this study, we systematically investigate the effect of different buoyancy terms in the Navier–Stokes equations via direct numerical simulations for flow inside the rotating container with an axial temperature gradient. Our results demonstrate that the buoyancy due to Coriolis forces and the inertia can change the flow behaviour significantly.

1The current study was supported by the ARC grant DP160103961. The computational resources provided by Pawsey under the grant D77 are greatly acknowledged.

5:55PM P13.00004 Boundary treatment for the compressible natural convection with discrete-unified gas-kinetic scheme, XIN WEN, U. of Delaware, LIAN-PING WANG, U. of Delaware and Southern U of Sci and Tech, ZHAOLI GUO, Huazhong U of Sci and Tech — The buoyancy-driven natural convection in an enclosure has been studied by researchers for several decades, it plays an important role in both flow physics study and practical applications. When the temperature difference is large, the flow is beyond the Boussinesq limit and governed by full compressible Navier-Stokes equations. An aspect that has not been well studied in the kinetic method is the boundary treatment for compressible thermal flow when the temperature field and velocity field are strongly coupled. How to implement the no-slip condition, the isothermal wall temperature and the adiabatic boundary condition can be a challenge for the kinetic method. In this talk, we propose a systematic approach of deriving the bounce-back boundary treatment for the straight boundary using the Chapman-Enskog expansion. The bounce-back expression allows the boundary nodes to satisfy the compressible Navier-Stokes equations. For a curved boundary, we implement the immersed boundary method into the discrete-unified gas-kinetic scheme (DUGKS). By design, the IB forces only contribute the leading order of momentum and energy equation, the no-slip condition and isothermal wall can then be implemented for the curved boundary.

6:08PM P13.00005 AFiD-MF: an efficient solver for three-dimensional multiphase flows, HAORAN LIU, QI WANG, KAI LEONG CHONG, Univ of Twente, ROBERTO VERZICCO, Univ of Rome, DETLEF LOHSE, Univ of Twente — We propose an extension of our code AFiD (www.afid.eu) to simulate efficiently three-dimensional multiphase flows. In this approach, we implement the phase field model into AFiD in order to retain the massive solver for the incompressible Navier-Stokes equations. The performance of AFiD has been confirmed in many previous studies. To simulate the dynamics of multiphase flows, we rely on the phase field model to capture the fluid-fluid interface and deal with large density/viscosity ratios of the phases. The coupling of the phase field model with the AFiD solver is obtained by the volume fraction distribution of each phase and the surface tension force on the fluid-fluid interface. Our new approach, AFiD-MF, is validated by comparisons with data in the literature and verified through several numerical experiments, such as an oscillating droplet, the deformation of a drop in shear flow, the breakup of a rising bubble and the Rayleigh-Bénard flow with two immiscible phases.
6:21PM P13.00006 Variational multiscale large eddy simulation with diffusive flux reconstruction for Rayleigh-Bénard convection. DAVID SONDAK, Institute for Applied Computational Science, Harvard University, JOHN SHADID, Department of Computational Mathematics, Sandia National Laboratories, TOM SMITH, Department of Computational Science, Sandia National Laboratories, SIDAFA CONDE, Department of Computational Mathematics, Sandia National Laboratories, ROGER PAWLOWSKI, Department of Computational Science, Sandia National Laboratories — Large eddy simulation (LES) models for Rayleigh-Bénard convection are developed using the variational multiscale formulation (VMS). In the VMS formulation, a sum decomposition is used to split the fields into resolved and unresolved components. The unresolved components are modeled to be proportional to the residual of the governing equations. In this way, if the numerical solution exactly captures all scales of motion, the residual will be zero and the unresolved portion of the field vanishes. The result is a consistent numerical method and a dynamic LES model. When discretizing the resolved scales with linear finite elements, the diffusive terms in the residuals are zero and the true residual is not satisfied. In Rayleigh-Bénard convection, this unbalanced residual may negatively impact results due to the importance of boundary layer effects. In the current work, the diffusive terms are reconstituted and included in the stabilized residual. This reconstruction is shown to provide better numerical convergence to the correct solution. Moreover, coarse near-wall resolution can be partially offset by correctly reconstructing the residual. All of these effects have a bearing on the scaling of the heat transport with Rayleigh number.

6:34PM P13.00007 A cross Criticality in the convection of Yield Stress fluids. FRANCESCA PELUSI, MAURO SBRAGAILLA, Department of Physics & INFN, University of Rome Tor Vergata, ANDREA SCAGLIERINI, MASSIMO BERNASCHI, Istituto per le Applicazioni del Calcolo CNR — We study numerically the Rayleigh-Bénard instability of a two-dimensional multi-component system confined between two horizontal walls heated from below and cooled from above. We first carefully validate the numerical model in the mixed regime, by studying the transition from conduction to convection in a homogeneous Newtonian system. As a further upgrade of complexity, the system is prepared in the de-mixed regime, with many liquid droplets closely packed together and separated by thin interfaces. In such conditions, the system is a yield stress fluid, i.e. it exhibits reference stress (the yield stress) below which it reacts to external perturbations as a solid, and above which it flows with non-Newtonian rheology. The transition to convection is characterized as a function of the packing and the intensity of the initial perturbation. When the system exhibits convection, a crucial additional feature with respect to the Newtonian homogeneous system is the presence of plasticity at the droplets scale, which is expected to alter the heat transfer from the hot to cold walls.

6:47PM P13.00008 Using machine learning to predict 1D steady-state temperature profiles from compressible mantle convection simulations. SIDDHANT AGARWAL, HEIBRiDS, German Aerospace Center (DLR), Technical University Berlin (TUB), NICOLA TOSI, DLR, TUB, DORIS BREUER, DLR, PAN KESSEL, GREGOIRE MONTAVON, TUB — Thermal evolution simulations of planetary mantles in 2D and 3D are computationally intensive. A low-fidelity alternative is to use scaling laws based on boundary-layer theory to express Nusselt Number (Nu) as a function of Rayleigh Number (Ra). Such a Ra-Nu relation can be used to run ‘0D’ parametrized evolution models by solving a simple energy balance equation. Yet scaling relations are available only for simple flows that cannot capture the full complexity of mantle dynamics. We propose leveraging Machine Learning to find a higher-dimensional mapping from five different parameters to the entire 1D temperature profile. The parameters are Ra, internal heating rate, dissipation number and the maximum viscosity contrast between top and bottom due to temperature and pressure. We train a Neural Network (NN) to take these inputs and predict the resulting steady-state temperature profile. The training data comes from a subset of 20,000 compressible simulations on a 2D cylindrical grid. This results in predictions with an average error of 1.6% on the test set. The NN can potentially be used to build a 1D evolution model by stacking several steady-state temperature profiles together, with each prediction serving as an input at the next time step.

Monday, November 25, 2019 5:16PM - 7:00PM — Session P14 Vortex Induced Vibration 307/308 - David Olson, Michigan State University

5:16PM P14.00001 A passive amplification of VIV through a semi-hollow cylinder for Reynolds number 200. SUNGMIN RYU, Incheon National University, SEUNGMIN KANG, Hanyang University. VIV CONTROL TEAM — Since the demonstration that the kinetic energy generated by vortex-induced vibrations (VIVs) can be converted into electricity, the amplification of VIV has been of significant interest. For this practical relevance, we present a passive method to amplify VIVs of a circular cylinder for Reynolds number 200. We employ an empty space inside a solid circular cylinder and place an orifice on the stagnation point to embody a concept of passive control, considering a potential combination with existing passive devices. We quantified the performance of the semi-hollow model via two-dimensional simulations for a range of reduced velocity

5:29PM P14.00002 Transverse vortex-induced vibrations of a prolate spheroid. ALVARO SANCHEZ RUIZ, Flying Whales / IMFT, REMI BOURGUET, IMFT / CNRS, GUILLAUME MARTINAT, Flying Whales — A prolate spheroid of aspect ratio 6:1 can be regarded as a simplified model of the hull of an aeronautical or underwater vehicle. From a fundamental perspective, it represents an intermediate geometry between a sphere and an infinite circular cylinder, which have often served as paradigms to study bluff body wakes and flow-induced vibrations. When a rigidly mounted, prolate spheroid is placed in a cross-current, i.e. a flow normal to its long axis, previous studies have shown that its unsteady wake is characterized by the formation of large-scale hairpin vortices, which result in fluctuations of fluid forces. The case where the spheroid is elastically mounted and the possible occurrence of flow-induced responses of the body remain to be explored. This is the object of the present work where the spheroid is free to vibrate in the transverse direction. On the basis of direct numerical simulations, the behavior of the coupled flow-structure system is examined, at Reynolds number 100, over a wide range of values of the oscillator natural frequency. Focus is placed on the emergence of vortex-induced vibrations and on the associated alteration of the flow dynamics, compared to the stationary body wake.
Combined vortex-induced vibration of a rigid cylinder and a detached flexible splitter-plate

Ehsan and Scanlan (1990) with the stationary (or steady) flow assumption. We thus propose a slight variant of sparse identification of nonlinear dynamics (Ehsan and Scanlan, 1990) with the stationary (or steady) flow assumption. The interaction of the plate and the cylinder vortices leads to plate tip-displacement enhancement at smaller S* and intermediate U*. The overall cylinder-plate response indicates maximum energy harvesting potential at high U* and low S*. The present work is the first study on the combined VIV and presents a promising system for energy harvesting.

A moving time window is introduced to apply a typical SINDy regression at each time instant during the VIV event. A library of candidate nonlinear functions is constructed based on the laboratory model. A rig allows the cylinder to vibrate with one degree-of-freedom giving rise to wake interactions. They are arranged in an equilateral triangle. The tests have been conducted on a section model elastically suspended for different angles-of-attack and spacing ratios. All results are based on self-excited structural responses for test cases of undamped and damped structures in the subcritical Reynolds no. range. Cross-flow Vortex/Wake-Induced Vibrations (VIV/WIV) in pure heave motion were identified. These were typically double responses resulting in a wide Strouhal no. range. Oscillation hysteresis contributed to the lowest onset. A special case related to torsion was also identified. The stability of the structure depends on the mass-damping parameter, the Scruton no. The Galloping-Induced Vibrations (GIV) are special cases identified because VIV and GIV couple when the Scruton no. is relatively low. When this coupling occurs, large vibrations develop. Thus, structural design, to include dampers, should assure that the Scruton no. is above a certain minimum value which has been identified to be substantially higher than the ones related to pure VIV/WIV cases.

 detachment of the cylinder is maintained using a computer-controlled air bearing rig. The cylinder is designed to have one degree-of-freedom mass-damping systems. The aero-elastic instability may lead to self-sustained oscillation due to the unsteadiness in the aerodynamic forces, induced by variation in the instantaneous angle of attack as the cable oscillates normal to the mean flow direction. In order to have better insight into the underlying flow physics, one-component molecular tagging velocimetry has been employed in this study to measure the boundary layer development on the cylinder with canonical geometry representing the suction line cross-section. Specifically, those cylinders have rectangular cross-section with thickness d, side ratio and corner radius of c/d = 2.5 and r/d = 0.5, respectively, and surface topology. Results will be presented for Reynolds numbers of ReD = 1100 and 2500, with focus on cross-stream profiles of the mean and the fluctuating streamwise velocity component for a range of angles-of-attack. Data will also be presented for baseline smooth-surface cylinders in order to investigate the effect of surface topology.

The present work is the first study on the combined VIV and presents a promising system for energy harvesting.

A special case relates to torsion was also identified. The stability of the structure depends on the mass-damping parameter, the Scruton no. The Galloping-Induced Vibrations (GIV) are special cases identified because VIV and GIV couple when the Scruton no. is relatively low. When this coupling occurs, large vibrations develop. Thus, structural design, to include dampers, should assure that the Scruton no. is above a certain minimum value which has been identified to be substantially higher than the ones related to pure VIV/WIV cases.

The Galloping-Induced Vibrations (GIV) are special cases identified because VIV and GIV couple when the Scruton no. is relatively low. When this coupling occurs, large vibrations develop. Thus, structural design, to include dampers, should assure that the Scruton no. is above a certain minimum value which has been identified to be substantially higher than the ones related to pure VIV/WIV cases.
5:16PM P15.00001 Reynolds stress spectra in pipes and boundary layers up to $Re_\tau \approx 10^4$\textsuperscript{1}. SPENCER ZIMMERMAN, JIMMY PHILIP, JOSEPH KLEWICKI, University of Melbourne — Turbulent pipe flows and zero-pressure-gradient boundary layers (BLs) are frequently compared with the goal of elucidating the role of boundary conditions on turbulent wall flows. Since at least the early study by Schubauer (J. Appl. Phys., 25 (2), 1954, pp. 188–196), differences in the Reynolds stress (RS) profiles of the two flows have been described as resulting primarily from the turbulent/non-turbulent intermittency of the boundary layer. More recently, however, a number of researchers have shown differences in the streamwise velocity variance contributions per scale between the two flows even below the intermittent range. Despite these differences, Monty et al. (J. Fluid Mech., 632, 2009, pp. 431–442) showed that the streamwise variance profiles of the two flows match over most of the domain at matched friction Reynolds number. Extending this work, Zimmerman et al. (J. Fluid Mech., 869, 2019, pp. 182–213) showed experimental evidence that differences in the wall-normal and spanwise RS profiles can extend to within the log-layer. Here, we use the same experimental dataset to discuss features of the spectra common to both flows as well as the scales and wall-positions at which systematic differences are observed.

\textsuperscript{1}Australian Research Council

5:29PM P15.00002 Spectral analysis of Reynolds shear stress in high Re turbulent channel flows\textsuperscript{1}. ROBERT MOSER, University of Texas at Austin, MYOUNGKYU LEE, Sandia National Laboratory — We perform spectral analysis of the terms in the transport equation Reynolds shear stress, $\langle u'v' \rangle$ in high $Re$ wall-bounded turbulent channel flows, using the analysis technique from Lee & Moser (J. Fluid Mech., vol 860, 886-938). Specifically, a log-polar representation of two-dimensional spectra are used to study the interactions of turbulence at different length-scales and wall-normal distances. The analysis results show that $\langle u'v' \rangle$ production occurs primarily in the streamwise-elongated modes. Inter-scale transfer at fixed wall-normal distances transfers shear-stress to modes that are elongated in the spanwise direction, especially away from the wall. Wall-normal transport then moves this streamwise-elongated stress to the other side of the channel. This exchange of stress between the two sides of the channel is the primary balance of the production, since the dissipation is relatively weak. Wall-normal transport of streamwise-elongated modes is more complex, with $\langle u'v' \rangle$ exchange driven by the increasing spanwise scale of the dominant stress carrying modes. Finally, away from the wall, the characteristic length scales of production and wall-normal transport mechanisms grow linearly with wall-normal distance.

\textsuperscript{1}This research used resources of the ALCF, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357.

5:42PM P15.00003 Identification of nonlinear interaction of the self-sustaining process in wall-bounded turbulence using resolvent analysis\textsuperscript{1}. H. JANE BAE, BEVERLEY MCKEON, California Institute of Technology — The nonlinear interaction in the self-sustaining process of wall-bounded shear flows is investigated using resolvent analysis. Resolvent analysis (McKeon & Sharma 2010) is used to identify the principal forcing mode which produces the maximum amplification in the minimal channel (Jiménez & Moin 1991). The identified mode is then removed from the nonlinear term of the Navier-Stokes equations at each time step from a direct numerical simulation of a minimal channel. The results show that the removal of the principal forcing mode is able to laminarize the flow, while the removal of subsequent modes only marginally affects the flow. Using conditional averaging, the flow structures that are responsible for generating the principal forcing mode, and thus the nonlinear interaction to self-sustain turbulence, are identified to be spanwise rolls interacting with meandering streaks.

\textsuperscript{1}This work was funded in part by the Coturb program of the European Research Council.

5:55PM P15.00004 Two-Scale Interaction in Near-Wall Turbulence. PATRICK DOOHAN, Imperial College London, ASHLEY P. WILLIS, University of Sheffield, YONGYUN HWANG, Imperial College London — It has been shown that the dynamics of individual energy-containing eddies in the hierarchy of wall-bounded turbulence (Townsend,1980) are governed by the self-sustaining process (SSP) (Hwang & Bengana,2016). However, multiscale flows also exhibit interaction between structures of different scales, notably the energy cascade, but the temporal dynamics of multiscale turbulence are not well understood. In this study, the temporal dynamics of a two-scale near-wall flow are investigated using a shear stress-driven model (Doohan et al.,2019). In addition to the SSP at each scale, the energy cascade and feeding from small- to large-scales are identified as the primary scale interaction processes. The energy cascade is most active during the streak-breakdown stage of the large-scale SSP and the timescale of the resulting small-scale dissipation matches that of the large-scale motion i.e. non-equilibrium cascade. The wall-normal cascade can also fuel small-scale production, driving the small-scale SSP. The feeding of large-scale structures is correlated with the streak-breakdown stage of the small-scale SSP and results in increased large-scale pressure transport and dissipation. In the presentation, the dynamics of the two-scale interaction system will be discussed in detail.

6:08PM P15.00005 Linear mechanisms sustaining wall turbulence\textsuperscript{1}. ADRIAN LOZANO-DURAN, Stanford University, MARIOS-A. NIKOLAIDIS, National and Kapodistrian University of Athens, MICHAEL KARP, Stanford University, NAVID C. CONSTANTINOU, Australian National University — Turbulence is the primary example of a highly nonlinear phenomenon. Nevertheless, there is evidence that the energy-injection mechanisms sustaining wall turbulence can be ascribed to linear processes. The different scenarios stem from linear stability theory and comprise modal instabilities from mean-flow inflection points, transient growth from non-normal operators, and parametric instabilities from temporal mean-flow variations, among others. These mechanisms, each potentially capable of leading to the observed turbulence structure, are rooted in simplified theories. Whether the flow follows any or a combination of them remains unclear. In the present study, we devise a novel collection of numerical experiments in which the Navier-Stokes equations are sensibly modified to quantify the role of the different linear mechanisms. This is achieved by direct numerical simulation of turbulent channel flows with constrained energy extraction from the streamwise-averaged mean-flow. We demonstrate that (i) transient growth alone is not sufficient to sustain wall turbulence and (ii) the flow remains turbulent when the modal instabilities are suppressed. On the other hand, we show that the parametric instability due to the time-varying mean-flow is essential to maintain turbulence alive.

\textsuperscript{1}Funded by by NASA under Grant #NNX15AU93A and ONR under Grant #N00014-16-S-BA10
6:21PM P15.00006 Statistical characterization of inter-component energy exchange in turbulent channel flows1. YONGSEOK KWON, JAVIER JIMENEZ, Universidad Politécnica de Madrid — In turbulent channel flows, it is well understood that the fluctuating velocity extracts energy from the mean flow via the lift-up mechanism (as represented by the production term in the energy budget equations). In this process, the streamwise velocity perturbation receives energy from the mean flow by the linear advection of the streamwise momentum along the cross-shear direction. However, what is still unclear is the precise mechanism by which the strong cross-shear velocity is triggered. In this presentation, this process is investigated from the perspective of the mean turbulent kinetic energy exchange and the inter-component energy exchange processes are statistically characterized. In particular, the primary focus is put on the non-linear interactions leading to the inter-component energy exchange, to shed light on their roles in the sustenance of turbulence in channel flows.

1Financially supported by the European Research Council through COTURB project, ERC-2014.AdG-669505

6:34PM P15.00007 Large field of view volumetric measurement of a turbulent boundary layer1. FELIX EICH, MATTHEW BROSS, Universität der Bundeswehr München, DANIEL SCHANZ, MATTEO NOVARA, ANDREAS SCHRÖDER, DLR Göttingen, CHRISTIAN J. KÄHLER, Universität der Bundeswehr München — In order to understand the complex dynamics and interaction processes between flow motions within turbulent boundary layers, time resolved 3D data is necessary. Therefore, a unique time resolved 3D measurement was performed at $Re_x = 42000$ under zero and adverse pressure gradient conditions. Using 13 high speed PIV cameras, helium filled soap bubbles as tracer particles in combination with LED illumination, it was possible to measure a turbulent boundary layer flow over $2.9 \times 0.6 \times 0.25 \text{m}^3$ at a recording frequency of 1 kHz. The acquired data was analysed with Lagrangian particle tracking techniques to generate velocity fields. The turbulent superstructures and their dynamics could be directly resolved. The interaction between the superstructures gives insight into the exchange of mass and momentum between the coherent flow motions. Furthermore the interaction between the superstructures with the flow separation line is studied. The results show that the superstructures have a significant effect on the dynamics of the line of separation.

1This work is supported by the German Research Foundation (Deutsche Forschungsgemeinschaft - DFG) KA1808/14 and SPP1881 Turbulent Superstructures KA1808/21 and SCHR 1165/5-1

6:47PM P15.00008 Streamwise development of targeted coherent structures in turbulent pipe flow. TYLER VAN BUREN, University of Delaware, LEO HELLSTROM, Princeton University, IVAN MARUSIC, University of Melbourne, ALEXANDER SMITS, Princeton University — Our aim is to perturb specific naturally occurring energetic modes in turbulent pipe flow ($Re_{\tau} = 3486$) using sections of pipe with a periodic non-circular cross-section. We track the downstream development of these perturbations with stereoscopic particle image velocimetry. Cross-sections with an azimuthally varying radius (sinusoidal) delicately agitate the flow through excitation of the Reynolds stresses. These sections significantly alter the mean flow, and add energy in the targeted structures while simultaneously reducing the energy in the non-excited structures. Supported under ONR Grant N00014-15-1-2402, Program Manager/Director Thomas and the Australian Research Council.

7:00PM P15.00009 Application of the attached eddy hypothesis for turbulence characterization in marine boundary layer flows1. DACHUAN FENG, The Hong Kong University of Science and Technology, VIKRANT GUPTA, MINPING WAN, Southern University of Science and Technology, LARRY K.B. LI, The Hong Kong University of Science and Technology — Tidal current turbines usually operate in moderate to high current marine boundary layer (MBL) flows. Whereas the mean flow speed largely determines the average power extraction, it is the higher-order turbulence statistics that determine the structural load on turbines (required for device design) and the wake length behind them (required for array design). We propose to use Townsend’s attached eddy hypothesis to characterize the turbulence in MBL flows. To this end, we perform large-eddy simulations of high-Reynolds-number MBL flows with seabed roughness varying from the transitional to the fully rough regime. We find that, within the log-layer, the horizontally-averaged spanwise turbulence intensity follows a log-linear law while the wall-normal component remains nearly constant. These findings are consistent with the attached eddy model (Townsend 1976, The Structure of Turbulent Shear Flow, CUP), with the constants being dependent on the seabed roughness. The present work provides a reduced-order framework for studying the effect of boundary layer turbulence on turbines.

1Shenzhen Science and Technology Program (Grant No. JCYJ20170412151759222), Research Grants Council of Hong Kong (Projects 16210418, 16235716 and 26202815)

Monday, November 25, 2019 5:16PM - 7:13PM –
Session P16 Focus Session: Exascale Computations of Complex Turbulent Flows III

5:16PM P16.00001 A massively parallel unstructured overset method for large-scale simulation of moving bodies in turbulent flows1. WYATT HORNE, KRISHNAN MAHESH, University of Minnesota — We present an unstructured overset method capable of performing direct numerical simulation (DNS) and large eddy simulation (LES) of many ($O(10^5)$) moving bodies, utilizing many computational cores ($O(10^5)$) as shown in Horne & Mahesh [J. Comput. Phys.(2019) In Press]. A dynamic overset assembly is conducted to connect mesh solutions. To establish communication patterns a parallel master/slave algorithm is used. A parallel flood-fill algorithm is used for cutting. For searches, k-d tree data structures are used. Often the connectivity between overset meshes remains the same between time steps. The temporal coherence of objects is directly used to only update necessary information with time, resulting in substantial cost savings. A non-dissipative finite volume method is used for the fluid flow. An interpolant is used which has superior kinetic energy properties compared to local reconstructions. To solve pressure, a penalty constraint formulation is used, resulting in a symmetric, positive definite system. Strong scaling is demonstrated for 100,000 particles in a turbulent channel flow up to 492,000 cores. Detailed flow results illustrating the method are presented.

1Supported by DOE
Objective of this work, therefore, is to explore the performance of multifidelity ensemble-based strategies in large-scale multiphysics applications. In this regard, multifidelity methods have become increasingly popular in the last years as acceleration strategies. Exascale computing resources promise to facilitate the use of these approaches on larger scale problems by providing 1-10k times augmented floating-point capacity. Significant fatigue load reduction is observed for all the simulated cases. The process is tested on a variety of test cases, including wall-resolved and wall-modeled LES of both canonical flows (channels and boundary layers) and complex memory hierarchies.

This work was supported by NASA.

1NASA Early Career Faculty Award (NNX15AU58G) and NSF (Award number: 1909379)

5:42PM P16.00003 High fidelity turbulence simulations with adaptive mesh refinement in Nek5000\(^1\), NICOLAS OFFERMANS, ADAM PEPLINSKI, PHILIPP SCHLATTER, Department of Mechanics, KTH Royal Institute of Technology, Sweden — The design of an adequate mesh is a complex and time-consuming task for users of computational fluid dynamics codes, which typically requires some a priori knowledge of the developing physics. We use adaptive mesh refinement, which combines error estimators and grid adaptation, as a tool for automatic mesh optimization. This strategy allows for better error control and easier mesh generation. Our framework is Nek5000, a highly-scalable code based on the spectral element method. The h-refinement technique is chosen for mesh adaptation, where selected elements are split via an octree structure, and two types of error estimators are considered. The first estimate relies on the local spectral properties of the solution. The second one is goal-oriented and based on the dual-weighted residual method, which requires the computation of an adjoint problem. Applications include direct numerical simulations of 3D turbulent cases such as the flow in a periodically constricted channel, also called periodic hill, and the flow around a NACA4412 wing profile at Re\(_c\) = 850,000. We ensure that the adaptive simulations are reliable and stable, and we compare the choice of error estimators on the refinement patterns and other relevant flow quantities.

1Knut and Alice Wallenbergs Foundation; SeRC Exascale Simulation Software Initiative (SESSI)

5:55PM P16.00004 Algorithmic grid selection in LES\(^1\), SIVASH TOOSI, JOHAN LARSSON, University of Maryland, IVAN BERMEJO-MORENO, University of Southern California — Given the recent progress in LES modeling and numerical schemes, the computational grid has now become the single most important factor determining the quality of an LES; and yet the current state-of-the-art is to rely fully on user expertise to build the grid. While this is a workable process for academic problems in relatively simple geometries, it becomes untenable going forward towards more complex flows in complex geometries with multi-physics effects at increasing computational scales. The present work is aimed at developing an algorithmic process for how to select a nearly optimal grid (maximal accuracy at minimal cost) for LES. Two error indicators are used to drive an iterative grid selection process, where the solution from a previous LES run is used to select a more optimal grid for a subsequent run. The resulting method is highly systematic, with minimal dependence on user input, and can be operated both in a free mode that results in unstructured grids and in a constrained mode that results in structured grids. The process is tested on a variety of test cases, including wall-resolved and wall-modeled LES of both canonical flows (channels and boundary layers) and more complex flows (backward-facing step and smooth-body separation) with excellent results in all cases.

1This work has been supported by NASA.

6:08PM P16.00005 Multifidelity ensemble-based prediction of turbulent flows at the Exascale\(^1\), LLUIS JOFRE, MANOLIS PAPADAKIS, ALEX AIKEN, GIANLUCA IACCARINO, Stanford University — The study of complex multiphysics turbulent flows is commonly based on intensive computational high-fidelity simulations. To build confidence and improve their prediction accuracy, very large computational budgets are typically required to characterize the impact of uncertainties on the quantities of interest. In this regard, multifidelity methods have become increasingly popular in recent years as acceleration strategies. Exascale computing resources promise to facilitate the use of these approaches on larger scale problems by providing 1-10k times augmented floating-point capacity, but at expenses of requiring more complex data management as memory is expected to become more heterogeneous and distributed. The objective of this work, therefore, is to explore the performance of multifidelity ensemble-based strategies in large-scale multiphysics applications using an Exascale-ready computational framework.

1This research was funded by the United States Department of Energy’s (DoE) National Nuclear Security Administration (NNSA) under the Predictive Science Academic Alliance Program (PSAAP) II at Stanford University, Grant DENA0002373.

6:21PM P16.00006 High-fidelity simulations for wind farm control co-design: evaluation of individual blade pitch control for turbine arrays and utility-scale wind farms\(^1\), FOTIS SOTIROPOULOS, XIAOLEI YANG, Stony Brook University, PETER SEILER, University of Minnesota — With the exponential growth of computer power, high-fidelity simulations are playing an increasingly important role enabling for the first time control co-design of wind farms, which can dramatically increase the annual energy production (AEP) and reduce the levelized cost of energy (LCOE). Individual blade pitch control (IBPC), which can effectively reduce the load fluctuations caused by the non-uniform incoming wind speed, has the potential to significantly reduce the LCOE of wind farms. IBPC, however, has been mostly evaluated for stand-alone individual wind turbines and its promise has yet to be demonstrated for turbine arrays especially at utility scale. We employ herein the VFS-Wind code to carry out large-eddy simulation (LES) with actuator-based parametrizations of turbine blades to explore the potential of IBPC in large wind farms. Two types of cases are investigated computationally: 1) a canonical turbine array with three different spanwise turbine spacings with the downwind turbine spacing fixed at seven rotor diameters; and 2) the XCEL Energy utility-scale wind farm in Pleasant Valley, Minnesota, United States, which consists of 100 wind turbines with generation capacity up to 200MW. Significant fatigue load reduction is observed for all the simulated cases.

1This work was supported by Xcel Energy through the Renewable Development Fund (RD4-13).
6:34PM P16.00007 Blade Resolved Wind Turbine Simulation with the Hybrid Time-Averaged Model Split Turbulence Model, JEREMY MELVIN, The Oden Institute, University of Texas at Austin, MARC HENRY DE FRAHAN, SHREYAS ANANTHAN, GANESH VIJAYAKUMAR, National Renewable Energy Lab (NREL), ROBERT MOSER, The Oden Institute, University of Texas at Austin — Blade resolved numerical simulations of wind turbines are an essential tool to help guide turbine design and placement in wind farms. Due to the high Reynolds number of these flows and the complex geometry of the turbine, a computational approach utilizing Reynolds Averaged Navier Stokes (RANS) or hybrid RANS/Large Eddy Simulation (LES) turbulence models are needed for computational feasibility. However, typical turbulence modeling approaches struggle with the complex flow characteristics present in flow fields around wind turbines. The newly developed Time-Averaged Model Split (TAMS) hybrid RANS/LES approach described by Haering et al. (AIAA Scitech 2019 Forum, AIAA 2019-0087, 2019) has shown the potential to resolve many of the issues with existing hybrid approaches for these complex flows. In this talk, we discuss efforts to conduct a blade resolved wind turbine simulation with the TAMS model. We integrate TAMS with a base SST RANS model, and conduct both airfoil and turbine simulations to compare performance and efficiency with standard DES hybrid approaches. We highlight the advantages of TAMS and discuss a path forward for additional improvements.

1 Exascale Computing Project (17-SC-20-SC), U.S. Department of Energy’s Office of Science and National Nuclear Security Administration

6:47PM P16.00008 Nek5000 LES of realistic urban geometries initialized from weather models, ALEKSANDR OABIKO, GKHAN SEVER, RAJEEV JAIN, YU-HSIANG LAN, Argonne National Laboratory, PAUL FISCHER, University of Illinois at Urbana-Champaign, HAOMIN YUAN, ROBERT JACOB, CHARLIE CATLETT, MUSIN MIN, Argonne National Laboratory — In the atmospheric modelling community, urban boundary layers have been generally treated using mesoscale models where a presence of obstacles are taken into account by parameterizations of the urban (or vegetation) canopy. There are not enough CFD-quality observations to validate the momentum and heat transfer in these models for all cases of interest. We seek to improve parametrization and complement observations with CFD reference solutions using an open-source CFD solver Nek5000. High scalability combined with high-order discretization and low count of degrees of freedom per processor allows an efficient exploitation of high-fidelity approaches like large-eddy simulation (LES) in complex geometries. With initial and boundary conditions derived from a High-Resolution Rapid Refresh (HRRR) initialized WRF-urban model, we have performed wall-resolved Nek5000 LES of realistic urban geometries including Lake Point Tower and Goose Island regions of Chicago. These preliminary simulations suggest we have the capability to start modelling the flow in any city geometry assuming a proper mesh can be built. Extending these reference simulations to larger domains will be possible with upcoming exascale supercomputers while longer simulation times may still remain a challenge.

1 Acknowledgement of the Exascale Computing Project (ECP) Urban and Center for Efficient Exascale Discretizations (CEED) funding and computational time allocations

7:00PM P16.00009 BCM-LES application to wind gust disaster under extreme meteorological events, TETSURO TAMURA, MASAHARU KAWAGUCHI, Tokyo Institute of Technology — Recently in Japan, people tend to realize so frequent occurrences of tornado and many attacks of typhoon to Japanese islands. In 2011, Tsukuba tornado arose in the Kanto plain based on a supercell. It causes the wind gust disaster on houses built on city area. In 2018, the typhoon Jebi attacked at Osaka area on September 4. High wind collapsed the wooden houses and the claddings of buildings and structures. This study applies LES based on BCM (Building Cube Method) to flow field for several km square area of disaster occurrence. This numerical model is formulated on the very fine Cartesian mesh system utilizing the IBM which can solve unsteady wind flows around actual complicated building shapes. Also, this research presents LES method for generating broad-banded turbulence flow that is able to regenerate high frequency components for existing meteorological model output. We performed the hybrid analysis of the meteorological model and engineering LES, in order to investigate near-ground turbulent wind under extreme meteorological events. This study numerically estimates the maximum wind velocity and the peak pressures on the building. Mechanism for process to failure of buildings will be discussed for establishment of safety at city.

Monday, November 25, 2019 5:16PM - 7:00PM — Session P17 Focus Session: Recent Advances in Data-Driven and Machine Learning Methods for Turbulent Flows IV

5:16PM P17.00001 Overview on sparsity in fluids, ZHE BAI, Lawrence Berkeley National Laboratory, STEVEN BRUNTON, University of Washington — Fluid flows are typically represented in high-dimension, although they often exhibit low-dimensional patterns. Understanding these patterns and their evolving dynamics is crucial for control. Thus, discovering these patterns from experimental and numerical data is a central challenge in fluid dynamics. The existence of these low-dimensional flow patterns also enable efficient sensing strategies, sparsity-promoting optimization, and randomized methods in fluids. In this talk, we will discuss integrated sparsification and modal decomposition for fluids, which involve compressed sensing, proper orthogonal decomposition, and dynamic mode decomposition, providing a foundation for pattern recognition and low-rank structure discovery of high-dimensional systems. These data-driven models save tremendous online experimental and computational resources by leveraging the existence of patterns. We will illustrate these ideas on a variety of engineering applications.

5:29PM P17.00002 Deep Neural Networks for Reduced Order Models for Fluid Flows, WILLIAM WOLF, HUGO LUI, University of Campinas — We present two numerical methodologies for construction of reduced order models, ROMs, of fluid flows through deep neural networks, DNNs. Here, the neural networks are used for regression and the frameworks are implemented in two contexts: one employs deep feedforward neural networks using a procedure similar to the sparse identification of non-linear dynamics algorithm, SINDy, and another is implemented using convolutional neural networks directly to the flow snapshots. The methods are tested on the reconstruction of a turbulent flow computed by a large eddy simulation of a plunging airfoil under dynamic stall. The reduced order models are able to capture the most energetic dynamics of dynamic stall including the leading edge stall vortex and the subsequent trailing edge vortex. The numerical framework allows the prediction of the flowfield beyond the training window and we demonstrate the robustness of the current ROMs constructed via deep neural networks through a comparison with sparse regression. The DNN approaches are able to learn transient features of the flow and present more accurate and stable long-term predictions compared to sparse regression.

1 Fundacao de Amparo a Pesquisa do Estado de Sao Paulo, FAPESP, under grant 2013/08293-7.
2 Mr. Hugo Lui may become the presenter in case he attends the meeting.
5:42PM P17.00003 Modeling particle-induced turbulence using sparse regression with embedded invariance¹. SARAH BEETHAM, JESSE CAPECELATRO, University of Michigan — Turbulence is ubiquitous in science and industry and is nearly always multiphase. Given current computational capabilities and the wide range of time- and length-scales of industrial systems, direct numerical simulation (DNS) is prohibitively costly. Thus, some degree of modeling must be employed. Current state-of-the-art modeling for turbulent multiphase flows is predominantly based on extensions to single-phase models, making it largely unsuccessful beyond the dilute limit. This eliminates the augmentation of existing models as an option for solving the multiphase closure problem. Our goal is to propose compact, tractable multiphase turbulence closures. Thus, we first derive the exact Reynolds Stress equations for multiphase flows, which identifies the specific terms requiring modeling. To arrive at these closures, we use sparse regression with embedded invariance. In this talk, we demonstrate the promise of this technique for three classes of flow with increasing difficulty: (1) single-phase free shear turbulence, (2) turbulent flow over periodic hills and (3) gas-solid sedimenting flow.

¹SB gratefully acknowledges support provided by the NSF Graduate Research Fellowship Program.

5:55PM P17.00004 Leveraging Dynamics for Near-Optimal, Ultra-Sparse Sensor Placement¹. SAMUEL OTTO, CLARENCE ROWLEY, Princeton University — Optimal sensor placement in high-dimensional nonlinear dynamical systems like fluid flows remains a challenging problem. Most current methods identify an overly large number of sensors because they do not make use of the time histories at each sensor location. Our work begins by constructing a POD subspace capturing the finite-time state trajectories of interest. The sensors must be able to robustly reconstruct trajectories in this subspace, leading to an objective function that has “nice” mathematical properties (namely, it is normalized, monotone, and submodular). These properties guarantee that an accelerated greedy algorithm for sensor placement has performance within a constant factor of the optimal performance. In addition to reconstructing trajectories in POD subspaces, our method can be extended to identify even fewer sensors that enable nonlinear reconstruction of trajectories on curved manifolds (which we call “ultra-sparse” sensor placement). We illustrate these methods with examples including a cylinder wake flow and Burgers equation.

6:08PM P17.00005 Space-time recovery of high-resolution turbulent flow fields with machine learning based super resolution. KAI FUKAMI, KOJI FUKAGATA, Keio University, KUNIHIKO TAIRA, University of California, Los Angeles — In recent years, the use of machine learning based super-resolution analysis has enabled accurate reconstruction of high-resolution image from its low-resolution counterpart. Moreover, machine learning techniques referred to as inbetweening have also been developed to estimate data in between temporal snapshots. Here, we combine two of these approaches to reconstruct complex multi-scale turbulent flows both in space and time. A convolutional neural network-based architecture called hybrid Downsampled Skip-Connection and Multi-Scale (DSC/MS) model is developed for the recovery of complex flow fields. The proposed model is applied to two-dimensional isotropic turbulence and three-dimensional turbulent channel flow at $Re_x = 180$ so as to demonstrate its capability in reconstructing spatio-temporal high-resolution turbulent flow fields from their coarse flow field data. We find that the present approach is able to accurately recover the high-resolution flow fields with only a modest amount of training data, despite the turbulent flow being complex and multi-scale in nature. The first two authors acknowledge the support by JSPS (18H03758). The last author thanks the support from ARO (W911NF-17-0118), and AFOSR (FA9550-16-1-0650).

6:21PM P17.00006 Reinforcement learning versus linear control of Rayleigh-Bénard convection, ALESSANDRO CORBETTA, GERBEN BEINTEMA, Eindhoven University Of Technology, LUCA BIFERALE, University of Rome, Tor Vergata, PINAKI KUMAR, FEDERICO TOSCHI, Eindhoven University Of Technology — Thermally driven turbulent flows are common in nature and in industrial applications. The presence of a (turbulent) flow can greatly enhance the heat transfer with respect to its conductive value. It is therefore extremely important -in fundamental and applied perspective- to understand if and how it is possible to control the heat transfer in thermally driven flows.

In this work, we aim at maintaining a Rayleigh-Bénard convection (RBC) cell in its conductive state beyond the critical Rayleigh number for the onset of convection. We specifically consider controls based on local modifications of the boundary temperature (fluctuations). We take advantage of recent developments in Artificial Intelligence and Reinforcement Learning (RL) to find automatically efficient non-linear control strategies. We train RL agents via parallel, GPU-based, 2D lattice Boltzmann simulations. Trained RL agents are capable of increasing the critical Rayleigh number of a factor 3 in comparison with state-of-the-art linear control approaches. Moreover, we observe that control agents are able to significantly reduce the convective flow also when the conductive state is unobtainable. This is achieved by finding and inducing complex flow fields.

6:34PM P17.00007 Active feedback control of flow over a circular cylinder with wall pressure sensor using machine learning¹. JINHYEOK YUN, JUNGIL LEE, Ajou University — In the present study, we conduct active feedback control of laminar and turbulent flows over a circular cylinder with wall pressure sensor for suppression of vortex shedding in the wake. The blowing and suction actuations are imposed at the wall before the flow separation, and their magnitudes are proportional to the transverse velocity in the the wake. To avoid direct measurement of velocity in the wake, we build an artificial neural network (ANN) between the pressures on the cylinder surface and the transverse velocities. For the learning process to build ANN, instantaneous flow data sets are obtained from numerical simulations of flow over a cylinder at $Re = 60$ and 3900. The performance of ANN is assessed with the locations of wall pressures, structures of neural network, and etc. It is found that the wall pressures on the cylinder surface can accurately predict velocities in the wake with the neural network built. Active feedback control combined with this neural network successfully suppresses the vortex shedding behind the cylinder, leading to reductions of the drag and lift fluctuations of cylinder.

¹Supported by the NRF Programs (2016R1D1A1B03932120 and 2019R1F1A1060841)
6:47PM P17.00008 Deep Learning for In-situ Compression of Large CFD Simulations

Monday, November 25, 2019 5:16PM - 7:13PM —
Session P18 Turbulence Theory: Dispersion and Stochastic Processes — 400 - Raul Bayoan
Cal, Portland State University

5:16PM P18.00001 On the stochastic modeling of Lagrangian velocity and acceleration in turbulent flows

5:42PM P18.00003 Linear Response Theory of Single Particle Diffusion in Turbulence

5:55PM P18.00004 The origin of turbulence in thermal convection.

6:08PM P18.00005 Fluctuation-induced forces in homogeneous isotropic turbulence

6:21PM P18.00006 Large-scale structure of a passive scalar field in homogeneous turbulence

6:47PM P17.00008 Deep Learning for In-situ Compression of Large CFD Simulations

, RYAN KING, ANDREW GLAWS, MICHAEL SPRAGUE, National Renewable Energy Laboratory — The ExaWind project seeks to develop blade-resolved LES simulations of wind turbines for next-generation exascale computing architectures. Such simulations are expected to generate data with over a billion degrees of freedom and upwards of a million time steps, requiring significant computational resources to be dedicated to data storage, visualization, and analysis. In many cases, performing these tasks on the full dataset is intractable, prompting the need for in-situ data compression. The simplicity of deep learning lends itself to this problem, with the promise of a nonlinear turbulent flow data. In this work, we explore deep learning methods for in-situ data compression, specifically a deep convolution autoencoder network that that maps 3D turbulent fields to a low-dimensional latent space. We compare the autoencoder against single-pass randomized SVD approaches in lossy restart studies where simulations are check-pointed and restarted. Our results show that an autoencoder trained on canonical turbulent flows can be applied to unseen configurations and is competitive with a single-pass SVD in terms of compression ratio, error, and computational cost.

On the stochastic modeling of Lagrangian velocity and acceleration in turbulent flows

, LAURENT CHEVILLARD, Laboratoire De Physique De l’ENS De Lyon, BIANCA VIGGIANO, Department of Mechanical and Materials Engineering, Portland State University, Portland, Oregon, USA, JAN FRIEDRICH, ROMAIN VOLK, MICHAEL BOURGOIN, Laboratoire De Physique De l’ENS De Lyon, RAUL BAYOAN CAL, Department of Mechanical and Materials Engineering, Portland State University, Portland, Oregon, USA — We propose to answer the following question: can we build up an infinitely differentiable stochastic process, such that asymptotically, when the Reynolds number goes to infinity, it becomes irregular (in a Holder sense) and intermittent (in a way we will clarify)? This has importance while modeling velocity and acceleration of particles following their trajectories in a turbulent flow. We propose such a process as a solution of a stochastic differential equation, making it causal. We proceed with analytical and numerical solutions, and compare against experimental and numerical data. Come, it will be fun.

Linear Response Theory of Single Particle Diffusion in Turbulence

, YUKIO KANEDA, Graduate School of Informatics, Nagoya University — A theory is proposed for the statistics of single particle diffusion in stationary homogeneous isotropic turbulence of incompressible fluid. The theory is based on a generalization of the idea of linear response theory that is known in the statistical mechanics for systems at or near the thermal equilibrium state. The theory gives \( \langle |\Delta v(t)|^2 \rangle / \langle v(t) \rangle = C_0 + B_1 + \ldots \), for the inertial time interval \( t \) such that \( T \gg t \gg \tau \), where \( \Delta v(t) \) is the velocity increment of a particle during the time interval \( t \), \( \tau \) is the kinetic-energy dissipation rate per unit mass, \( T \) is the integral time scale, \( \tau \) is the Kolmogorov micro-time-scale, \( B_1 \) is a non-dimensional universal constant. \( C_0 \), and \( C_1, C_T, C_\tau \) are non-dimensional universal constants.

The origin of turbulence in thermal convection.

, KATEPALLI SREENIVASAN, New York University, JOERG SCHUMACHER, AMBRISH PANDEY, Technical University, Ilmenau, Germany, VICTOR YAKHOT, Boston University — If a fluid flow is driven by a weak Gaussian random force, the nonlinearity in the Navier-Stokes equations is negligibly small and the resulting velocity field obeys Gaussian statistics. Nonlinear effects become important as the driving becomes stronger and a transition occurs to turbulence with anomalous scaling of small scales. This process is reasonably well understood for homogeneous and isotropic turbulence. In this paper, we discuss the Reynolds-number dependence of moments of the kinetic energy dissipation rate in the bulk of thermal convection in the Rayleigh-Benard system. The data for different Reynolds numbers are obtained from direct numerical simulations using the dimensionless spectral element method in a convection cell with square cross section and aspect ratio 25. The normalized moments of the kinetic energy dissipation rate show a non-monotonic dependence for small Reynolds numbers before obeying the algebraic scaling for the turbulent state. This feature is explained via the transition akin to “soft-to-hard turbulence” in convection where, depending on the Rayleigh number, turbulence is produced by either the instabilities of the bulk and the wall boundary layers.

Fluctuation-induced forces in homogeneous isotropic turbulence

, RODOLFO OSTILLA MONICO, DANIEL PUTT, University of Houston, VAMSI ARZA SPANDAN, Harvard University, ALPHA ALBERT LEE, University of Cambridge — Understanding force generation in non-equilibrium systems is a significant challenge in statistical physics. We uncover a surprising fluctuation-induced force between two plates immersed in homogeneous isotropic turbulence using Direct Numerical Simulation. The force is a non-monotonic function of plate separation. The mechanism of force generation reveals an intriguing analogy with fluctuation-induced forces: energy in the fluid is localised in regions of high vorticity, or “worms”, which have a characteristic length scale.

Large-scale structure of a passive scalar field in homogeneous turbulence

, KATSUNORI YOSHIMATSU, YUKIO KANEDA, Nagoya University — We study the large-scale structure of a passive scalar field without any scalar source in incompressible homogeneous turbulence. We assume that the initial scalar spectrum at time \( t = 0 \) takes the form \( Ck^2 + o(k^2) \) at the wavenumber \( k \to 0 \), where \( C \) is independent of \( k \). Theoretical analysis [Yoshimatsu and Kaneda, Phys. Rev. Fluids 3, 104601 (2019)] shows that the spectrum keeps the form at \( k \to 0 \) and \( C \) is time independent for \( t \geq 0 \). On the basis of the independence and a constant assumption of a certain self-similar evolution of the scalar field at the large scales including the scales comparable to the scalar integral length scales, it is shown that a certain measure of the anisotropy of the scalar field remains time-independent at the large scales in a self-similar state, irrespective of the velocity field. In addition, we performed direct numerical simulation (DNS) of the passive scalar field in a periodic box. It is found that the DNS results are consistent with the theory.
6:34PM P18.00007 Three-point statistics of passive scalars at high Schmidt numbers\textsuperscript{1}, M. P. CLAY, Georgia Institute of Technology, K. P. IYER, D. BUARIA, New York University, P. K. YEUNG, Georgia Institute of Technology, K. R. SREENIVASAN, New York University — The turbulent mixing of passive scalars is a fundamental problem relevant to many natural and engineering flows. While traditionally analyzed via one- or two-point statistics, three-point statistics have also been used to gain insight into the structure of the scalar field [Warhaft, Annu. Rev. Fluid Mech 32, 203–240 (2000)]. Experimental data are scarce, and for the important case of scalar fluctuations generated under the presence of a mean gradient in isotropic turbulence, measurements are limited to Schmidt numbers (Sc) near unity [Mydlarski and Warhaft, Phys. Fluids 10, 2885–2894 (1998)]. Here we analyze three-point statistics from direct numerical simulations of scalars under a uniform mean gradient in $R_{\lambda}$ ≈ 140 forced isotropic turbulence. By using grids with up to 8192\textsuperscript{3} points and passive scalars with Sc up to 512, three-point statistics are gathered in the emerging viscous-convective range to study the approach to local isotropy exhibited by high-Sc scalars.

\textsuperscript{1}Supported through supercomputer resources at OLCF (DOE INCITE 2017) and TACC (XSEDE).

6:47PM P18.00008 Triad interactions in a four-dimensional fluctuating velocity field, PREBEN BUCHHAVE, Intarsia Optics, CLARA VELE, Technical University of Denmark — High intensity turbulence should be considered a stochastic function of four independent parameters, three spatial coordinates and time. The interaction between the different velocity structures is caused by the nonlinear term in Navier-Stokes equation and is conventionally analyzed by a three-dimensional spatial Fourier transform in a homogeneous velocity field resulting in the so-called triad interactions caused by the second order convection term. We present some conclusions resulting from an analysis of a four-dimensional Fourier transform of the fluctuating velocity. Moreover, we consider a case corresponding to a realistic experiment, where the velocity is digitally sampled with a finite sample rate and the flow is limited to a finite spatial and temporal range. Our results explain features observed in experiments such as the time development of the power spectrum, deviations from the conventional Richardson cascade and perseverance of initial large-scale spectral features.

7:00PM P18.00009 Triple-Correlations in Decaying Isotropic Turbulence\textsuperscript{1}, CLAYTON BYERS, Trinity College, JONATHAN MACART, University of Illinois at Urbana-Champaign, MICHAEL MUELLER, MARCUS HULTMARK, Princeton University — The self-similar scaling approach for decaying isotropic turbulence is utilized to extract constraints on the temporally dependent scaling parameters. The resulting similarity solution and constraints show that the temporal evolution of the length scale, which is shown to be the Taylor microscale, sets the exponent of decay. This exponent is found to retain a dependence on the initial conditions of the flow. Additionally, a new triple-correlation scaling parameter, $u'\lambda/Re_{\lambda}$, is found. The validity of this new scaling is checked in three different ways, each resulting in a collapse of the triple correlation data. The usefulness of this scale becomes apparent when compared to the results of Stewart & Townsend (1951), which utilized the classic $u'^{3}$ scaling and did not show collapse in their data. Three Direct Numerical Simulations at differing initial Reynolds numbers were performed to test the theoretical results.

\textsuperscript{1}The authors gratefully acknowledge valuable support in the form of computational time on the TIGRESS high performance computer center at Princeton University, which is jointly supported by the Princeton Institute for Computational Science and Engineering (PICS:IE) and the Princeton University Office of Information Technology’s Research Computing department. Portions of this work were supported by the NASA CT Space Consortium Subaward P-1316.

Monday, November 25, 2019 5:16PM - 7:00PM –
Session P19 Advanced Turbulence Models II 401 - Guillaume Blanquart, Caltech

5:16PM P19.00001 Colouring turbulence with the nonlinear terms\textsuperscript{1}, SEAN SYMON, SIMON ILLINGWORTH, IVAN MARUSIC, University of Melbourne — We study the behaviour of the nonlinear terms in turbulent channel flow from direct numerical simulations (DNS) at low and moderate Reynolds numbers. These terms correspond to the nonlinear forcing of the (linear) resolvent-based models of McKeon & Sharma 2010. Spectral proper orthogonal decomposition (SPOD) is used to extract the most energetic structures and identify the power spectrum of the most energetic modes. We also compare the nonlinear terms to the equivalent nonlinear forcing supplied by eddy viscosity-enhanced resolvent models. At spatial wavenumbers where there is energetic activity, the nonlinear terms share many similarities with their velocity fluctuation counterparts. The nonlinear terms are also a good match with predictions from eddy viscosity at wavenumber pairs where the resolvent operator including eddy viscosity is low-rank. The implications for estimation techniques such as spectral linear stochastic estimation and the Kalman filter will be discussed.

\textsuperscript{1}Australian Research Council

5:29PM P19.00002 3D diffuser flow predictions using lag models, RAJARSHI BISWAS, Iowa State University, PAUL DURBING, Professor, Iowa State University — Lag RANS models employ the usage of a scalar parameter designed to represent the stress-strain misalignment in turbulent flows. The parameter is used to scale the eddy viscosity particularly in the near-wall region. This proves effective for improved prediction of 2D flows with separation. Two novel models, Lag k-epsilon and Lag k-omega are tested for a 3D diffuser configuration. The geometry is parameterized by the inlet aspect ratio(AR). LES predictions show the separation bubble switches position from one wall to the other as AR is increased. Typical RANS formulations such as k-omega SST predict the flow field inaccurately. The lag models are tested for six different ARs and compared against commonly used RANS models.

\textsuperscript{1}Office of Naval Research, grant #N00014-17-1-2200

5:42PM P19.00003 Linear Navier-Stokes based model for turbulent channels with unstable stratification, ANAGHA MADHUSUDANAN, SIMON ILLINGWORTH, IVAN MARUSIC, University of Melbourne — Studies have shown that the linearized Navier-Stokes equations model the coherent large-scale structures in turbulent wall-bounded flows reasonably well. In the present work we aim to understand if this linear model can be extended to study the coherent large-scale structures that have been experimentally and numerically observed in turbulent Rayleigh–Bénard–Poiseuille flows [e.g., Chauhan et al., 2013, Pirozzoli et al., 2017]. In particular, we concentrate on two features of these structures. First, we look at the wall-normal coherence of the streamwise constant modes. And second, we study the inclination angle of the large-scale structures in these flows. These features are then compared to the available results from numerical and experimental studies.

References
5:55PM P19.00004 Restricting integral length scale growth in triply periodic turbulence simulations, LIMBERT PALOMINO, CHANDRU DHANDAPANI, JEFF RAH, GUILLAUME BLANQUART, California Institute of Technology — The 3D periodic box is an essential tool for studying turbulence. It is both an appropriate canonical configuration for homogeneous isotropic turbulence and a computationally efficient configuration to simulate. Unfortunately, without an active mean of generating turbulence, the turbulent kinetic energy decays over time due to viscous dissipation. Through the years, various methods of forcing the Navier-Stokes have been proposed to maintain this statistically stationary turbulence, including spectral and linear forcing. Although linear forcing schemes fully capture the physics of turbulence, as the simulation evolves in time, the largest eddies in the simulation grow to the order of the computational domain size. The current study characterizes this growth in terms of both the integral length scale and the corresponding energy spectra. Furthermore, we propose a modified linear forcing technique that is analogous to a re-scaling of the computational domain at each time step. This provides more active control over the integral length scale and eddy growth in the simulations.

6:08PM P19.00005 The Macroscopic Forcing Method and incorporating non-locality into macroscopic models1, JESSIE LIU, ALI MANI, Stanford University — The Macroscopic Forcing Method (MFM) (Mani and Park (2019), arXiv:1905.08342) is a newly-developed technique that can be used to determine differential operators associated with turbulence closure. The results of MFM can then be used to improve macroscopic models, i.e. models that describe averaged quantities such as Reynolds-averaged Navier-Stokes (RANS) models. One issue is that often the found differential operators are non-local and may be difficult to incorporate into existing models. We present application of MFM to an example problem involving scalar transport and temporal non-locality. We then present a method for easily incorporating the effects of temporal non-locality into the macroscopic model for the averaged-scalar transport.

6:21PM P19.00006 On new symmetry-generated RANS models1, DARIO KLINGENBERG, Graduate School of Excellence Computational Engineering, TU Darmstadt, MARTIN OBERLACK, DOMINIK PLUEMACHER, Chair of Fluid Dynamics, TU Darmstadt — We apply new insights into turbulent statistics obtained through Lie-symmetry analysis to the problem of RANS modeling. Symmetries mirror key physical principles from governing equations. The symmetries of the infinite hierarchy of multi-point correlation equations fall into two categories. Symmetries of classical mechanics, which have direct counterparts in the unaveraged Navier-Stokes equations, and statistical ones, which only arise when adopting a statistical view on turbulence. It was shown that some of the latter type encode crucial phenomena, in particular intermittency and non-Gaussianity (Waclawczyk 2014). In a modeling context, symmetries provide constraints on the model equations, because unless the model equations contain precisely the same symmetries as the exact equations, the model is prone to exhibiting nonphysical behavior. We therefore present a new modeling framework that allows algebraically generating turbulence models based on symmetries. We apply this method to obtain a prototype RANS model that is agreement with not only the classical, but also the statistical symmetries, which existing models fail to accomplish. It turns out to be necessary to introduce as a new model variable an additional velocity field with specific advantageous symmetry properties.


6:47PM P19.00008 Multi-level stochastic refinement for turbulent time series and fields, MICHAEL SINHUBER, Max Planck Institute for Dynamics and Self-Organization, Gottingen, JAN FRIEDRICH, ENS Lyon, RAINER GRAUER, Ruhr-Universitaet Bochum, GREGORY P. BEWLEY, Cornell University, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization, Gottingen — Many high-dimensional systems exhibit complex spatio-temporal dynamics. Typically, this complexity comes along with strong multi-scale correlations and scale-dependent deviations from Gaussianity, which requires multi-time-multi-point statistics for a full characterization. In many practical cases, obtaining sufficiently resolved data is prohibitively expensive or even impossible. While a statistical characterization of such systems is often sufficient, many applications where turbulence plays a key role require a complete spatio-temporal realization of the system. Examples range from modeling particle propagation in fusion plasmas to wind field modeling for wind energy applications. To address this challenge, we develop a stochastic refinement method that generates finely resolved data sets from readily available, coarsely sampled turbulence data as well as synthetic datasets within the scope of classical turbulence models. This is done by utilizing scale-Markovian properties of turbulence and scale-dependent three-point probability statistics. We test our approach both for wind tunnel data from the VDTT at the Max-Planck-Institute for Dynamics and Self-Organization in Gottingen as well as for simulated turbulent fields based on classical turbulence models.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P20 Experimental Techniques: Data Analysis, Bias and Uncertainty 602 - Barton Smith, Utah State University
compared to the change in voxel size. The bias error estimate is proportional to the square of the noise's standard deviation. Respectively, the estimates were compared to the true bias error calculated using the true velocity profile. Results show that the bias error is for measurement error, but neglects bias error. This study estimates bias error by integrating an approximated intravoxel velocity profile within providing time-resolved, volumetric, 3-directional velocity information of a patient's cardiovascular flow in vivo. This technique suffers from of high importance in space flight. However, the combustion process in hybrid rocket engines is still a matter of ongoing research and not fully understood yet. Recently, combustion tests with different paraffin-based fuels have been performed at the German Aerospace Center (DLR). For a better understanding of the experiments, the combustion process has been captured with a high-speed video camera, which leads to a huge amount of images for each test. In order to catch the essential flow structures, the combustion dataset has been analyzed with unsupervised machine learning techniques. In this talk, we present the outcome of the clustering. Using the machine learning techniques, valuable insights into the different combustion phases were obtained and a comparison of the quality of the combustion flame in the different tests could be made. In particular, depending on the fuel formulation and oxidizer mass flow, differences in the transients and flame brightness were found.

This work highlights the fact that the naive application of ML/AI will generally be insufficient to extract universal physical laws without further modification.

5:29PM P20.00002 Data Analysis of Hybrid Rocket Fuels Combustion Tests ALEXANDER RUETTGERS, German Aerospace Center (DLR), Simulation and Software Technology, ANNA PETRAROLO, MARIO KOBALD, German Aerospace Center (DLR), Institute of Space Propulsion — Clustering techniques were applied to hybrid rocket combustion tests to better understand the complex flow phenomena. Novel techniques such as hybrid rockets that allow for cost reductions of space transport vehicles are of high importance in space flight. However, the combustion process in hybrid rocket engines is still a matter of ongoing research and not fully understood yet. Recently, combustion tests with different paraffin-based fuels have been performed at the German Aerospace Center (DLR). For a better understanding of the experiments, the combustion process has been captured with a high-speed video camera, which leads to a huge amount of images for each test. In order to catch the essential flow structures, the combustion dataset has been analyzed with unsupervised machine learning techniques. In this talk, we present the outcome of the clustering. Using the machine learning techniques, valuable insights into the different combustion phases were obtained and a comparison of the quality of the combustion flame in the different tests could be made. In particular, depending on the fuel formulation and oxidizer mass flow, differences in the transients and flame brightness were found.

5:42PM P20.00003 Data assimilation method to de-noise and de-filter particle image velocimetry data. ROLAND BOUFFANAIS, Singapore University of Technology and Design, JURRIAAN J. J. GILLESSEN, University College London, DICK K. P. YUE, Massachusetts Institute of Technology — We present a variational data assimilation method in order to improve the accuracy of velocity fields \( \tilde{v} \), that are measured using particle image velocimetry (PIV). The method minimizes the space-time integral of the difference between the reconstruction \( u \) and \( \tilde{v} \), under the constraint, that \( u \) satisfies conservation of mass and momentum. We apply the method to synthetic velocimetry data, in a two-dimensional turbulent flow, where realistic PIV noise is generated by computationally mimicking the PIV measurement process. The method performs optimally when the assimilation integration time is of the order of the flow correlation time. We interpret these results by comparing them to one-dimensional diffusion and advection problems, for which we derive analytical expressions for the reconstruction error.

This work is partially funded by the SUTD-MIT International Design Center

5:55PM P20.00004 State-space Optimized Dynamic Mode Decomposition for Noisy Data. TAKU NONOMURA, Tohoku University, Presto, JST, KAZUYUKI NAKAMURA, Meiji University, Presto, JST, NAOTO NAKANO, Kyoto University, STEVEN L. BRUNTUN, J. NATHAN KUTZ, University of Washington — This presentation proposes several new formulations of dynamic mode decomposition (DMD) for full-state measurements of a linear dynamical system with process and measurement noise. First, we develop two methods to denoise and reconstruct the true state of the approximated linear system from noisy experimental data: the DMD-based state variable reconstruction (DMDsrv) and the DMD-based state space reconstruction (DMDssr). DMDsrv estimates the state variable as a solution of a least square problem, when the system coefficients and the noise variances are known. DMDssr simultaneously estimates the noise variance and the state variables by the expectation-maximization (EM) algorithm in a Bayesian framework. The final method, state-space optimized DMD (soDMD), simultaneously estimates the DMD coefficients together with the noise variances and the state variable. The proposed soDMD can estimate the system coefficients, noise variance and true state variables from noisy data. Estimation of system coefficients and noise variances can be used for data-assimilation using Kalman filter. Numerical tests show an improvement of the proposed methods over conventional DMD for linear systems with process noise.

This research is partially supported by Presto, JST (JPMJPR1678) and IADF.

6:08PM P20.00005 4D Flow MRI Bias Error Estimation. SEAN ROTHENBERGER, Weldon School of Biomedical Engineering, Purdue University, JIACHENG ZHANG, MELISSA BRINDISE, School of Mechanical Engineering, Purdue University, SUSANNE SCHNELL, Feinberg School of Medicine, Northwestern University, PAVLOS VLACHOS, School of Mechanical Engineering, Purdue University, VITALIY RAYZ, Weldon School of Biomedical Engineering, Purdue University — 4D flow MRI is a non-invasive imaging technique providing time-resolved, volumetric, 3-directional velocity information of a patient’s cardiovascular flow in vivo. This technique suffers from low VNR and resolution. ‘Enhanced’ 4D flow MRI measurements can be produced by informing the measurements with high-resolution flow modeling methods, e.g. CFD, PIV. However, the error of the 4D flow MRI measurements must first be defined. Past literature defines a model for measurement error, but neglects bias error. This study estimates bias error by integrating an approximated intravoxel velocity profile within the limits of the voxel dimensions. The model of bias error was tested in synthetic flow with a velocity range of 2m/s. The effect of the voxel size and noise was determined by investigating a range of voxel sizes and noise standard deviations of 1 to 0.147mm³ and 0 to 6cm/s, respectively. The estimates were compared to the true bias error calculated using the true velocity profile. Results show that the bias error is proportional to the square of the noise’s standard deviation.

This work is partially supported by Presto, JST (JPMJPR1678) and IADF.

NIH R21 NS 106696 award
6:21PM P20.00006 Unscented Kalman filter (UKF) based nonlinear parameter estimation for a turbulent boundary layer: a data assimilation framework. ZHAO PAN, University of Waterloo, YANG ZHANG, JONAS GUSTAVSSON, Florida State University, JEAN-PIERRE HICKEY, University of Waterloo, LOUIS CATTAFASTA, Florida State University, UWATERLOO COLLABORATION, FSU COLLABORATION. - There is a vast amount of data available for the turbulent boundary layer and its potential use for predictive purposes has been established recently. However, accurate measurements of turbulent boundary layer parameters (e.g., friction velocity $u_*$ and wall shear $\tau_w$), are challenging, especially for high speed flows. Many direct and/or indirect diagnostic techniques have been developed to measure wall shear stress. However, based on different principles, these techniques usually give different results with different uncertainties. The current study introduces a nonlinear data assimilation framework based on the Unscented Kalman Filter that can fuse information from 1) noisy and gappy measurements from Stereo Particle Image Velocimetry, a Preston tube, and a MEMS shear stress sensor, as well as ii) the uncertainties of the measurements to estimate the parameters of a turbulent boundary layer. A direct numerical simulation of a fully developed turbulent boundary layer flow at Mach 0.3 is used first to validate the data assimilation algorithm. The algorithm is then applied to experimental data of a flow at Mach 0.3, which are obtained in a blowdown wind tunnel facility. The UKF-based data assimilation algorithm is robust to uncertain and gappy experimental data and is abl

6:34PM P20.00007 Power Law Decay Estimation for Turbulent Spectral Densities. CARL R. HART, U.S. Army Engineering Research and Development Center, Cold Regions Research and Engineering Laboratory, GREGORY W. LYONS, U.S. Army Engineer Research and Development Center, Construction Engineering Research Laboratory, NATHAN E. MURRAY, The University of Mississippi, National Center for Physical Acoustics — Turbulent flows commonly feature power law decay in one or more field quantities, such as the $-5/3$ inertial subrange power law for velocity spectra. Assuming sufficient time series data are collected, the problem of estimating the power law decay rate of a turbulent spectral density relies on two factors: the correct choice of data window in statistical signal processing, and an objective procedure to estimate the power law decay rate. In this context the single most important factor for a data window is the side-lobe decay rate. Ensuring the side-lobe decay rate exceeds that of measured data avoids the subtle error of spectral leakage. An objective procedure to estimate the power law decay rate is based on a maximum likelihood estimator. Under the assumption that the Fourier transform of turbulence time series is a circularly-symmetric complex normal random variable a likelihood function for the power spectral density is based on a Gamma distribution. Maximizing the log-likelihood, with a spectral model that parameterizes the Gamma distributions, leads to a robust estimator for the power law decay rate. These concepts are illustrated through synthetic realizations of colored noise, acoustic measurements of a supersonic turbulent jet, and atmospheric surface-layer turbulence.

6:47PM P20.00008 Identification of Dynamic Atmospheric Conditions via Total Variation. NICHOLAS HAMILTON, National Renewable Energy Laboratory — Selection of atmospheric events that conform to particular conditions of interest within multivariate data is necessary to validate emerging high-fidelity simulations of wind plant flows. Conditions of interest are frequently determined simply as those that occur most often, given the need for well-converged statistics from observations. Aggregation of observations without regard to covariance between time series discounts the dynamical nature of the atmosphere and is not sufficiently representative of wind plant operating conditions. Identification and characterization of continuous time periods representative of atmospheric conditions that have a high value for analysis or simulation sets the stage for validation of more advanced physical mechanisms. The total variation of the atmosphere is a metric that takes into account variability within each channel as well as covariance between channels and identifies periods of interest that conform to desired objective functions, such as quiescent conditions, wind speed ramps or waves, or sudden changes in wind direction. Direct identification and classification of events of interest within atmospheric data sets is vital to developing our understanding of wind plant response and to the formulation of forecasting and control models.

7:00PM P20.00009 Using Machine Learning to Determine the Velocity Information Content in OH-PLIF Images. SHIVAM BARWEY, MALIK HASSANALY, VENKAT RAMAN, University of Michigan, ADAM STEINBERG, Georgia Institute of Technology. — This study determines the velocity field information contained purely in OH-PLIF images in the closed domain of a premixed swirl combustor. A fully convolutional neural network (CNN) is used with a dataset containing simultaneous OH-PLIF and PIV measurements in both attached and detached flame regimes. To facilitate the study, the CNN represents a direct projection from OH-PLIF to PIV field. Two types of models are trained: 1) a global CNN which is trained using images from the entire domain, and 2) a set of local CNNs which are trained only on individual sections of the domain. Local models show improvement in creating PIV fields in both attached and detached regimes over the global models in most settings. A comparison between model performance in attached and detached regimes shows that the CNNs are much more accurate across the board in creating velocity fields for attached flames. Further, time history inclusion in the OH fields is also studied, as is the ability of the model to extrapolate to unexplored regions of the domain. Ultimately, this work shows that there is redundant information in the OH-PLIF images, and can open the door for the development of diagnostic tools that decrease the overlapping content between simultaneously measured fields.


5:16PM P21.00001 Pressure and Shear Stress Distribution of Drop Impacts. TING-PHI SUN, XIANG CHENG, University of Minnesota, Twin Cities — Drop impacts are ubiquitous and relevant to many important natural and industrial processes. Although the kinematics of drop impact such as the morphology of impactor drops have been extensively studied experimentally due to the fast advance of high-speed photography techniques, the dynamic aspects of drop impacts remains largely unexplored. Here, we investigate the pressure and shear stress distributions of drop impacts via a newly-developed experimental tool, high-speed 3D stress microscopy. By combining laser-sheet illumination with high-speed photography, we track the fast movements of fluorescent particles embedded in elastic gels under the impact of liquid drops. The measurements enable us to obtain a strain of the elastic gels induced by the impact. The temporal evolution of impact pressures and shear stresses of liquid drops can then be extracted based on the strain-stress relation of continuum mechanics. Our study on the pressure distribution confirms the key prediction of the self-similar theory and numerical simulations, where the maximum impact pressure occurs near the contact line, rather than the center of impacting drops. In addition, we also quantify the fast temporal evolution of impact-induced shear stresses, information crucial for mitigating impact-induced damages on solid substrates. This research is support by NSF CAREER DMR-1452180.
reduce the degree of non-monotonicity of the transitional boundaries between bouncing and merging states.

reported for single-liquid systems, a third kind of merging was also observed for two-liquid systems. This third kind of merging was found to

on liquid film using two liquids with similar surface tension, but different viscosities. The results with two-liquid systems show a shift in the

of different liquids with varying properties. Recognizing its importance, in this talk, we will present a study on the dynamics of drop impact

technologies such as 3D inkjet printers can print multiple materials, very often the drop and the impacted the liquid film are required to be

from merging to bouncing, back to merging and then to bouncing again, as we increase the film thickness. Since many of the advanced printing

film showed that the impact could can result in two outcomes, namely bouncing and merging, and the transition between these two states is

is critical in several industrial applications, including inkjet printing and thermal sprays. Previous studies using single liquid for both drop and

features through experiments and theoretical models.

Rainfall on biological superhydrophobic surface (e.g. bird feathers, insect wings, plant leaves, etc) is ubiquitous in nature. Previous studies in

contribution to the volume of the drop that penetrates the porous slice. We provide criteria based on our experimental measurements that can

be used to predict when inertia driven flow into porous materials will contribute significantly to the total volume of liquid that penetrates the

porous material.

Shock wave-induced drop fragmentation upon raindrop impact

on biological surfaces , SEUNGHO KIM, BRIAN WU, JASON J. DOMBROWSKIE, SUNGHWAN JUNG, Cornell University — Rainfall on biological superhydrophobic surface (e.g. bird feathers, insect wings, plant leaves, etc) is ubiquitous in nature. Previous studies in the laboratory have focused on low-speed impacting drops (less than 1 m/s) only, which is far from the speed of real raindrops (more than 5 m/s). In this present work, we explore raindrop impact at high speeds, which exhibits unexpected drop dynamics: numerous shock-like waves are generated on a spreading drop in the presence of microscopic textures on biological surfaces. Then, the spreading drop with shock-like waves is fragmented soon after it approaches a maximal spreading extent, thereby reducing the residence/contact time more than twofold. Since it is known that the heat and momentum transfer of an impacting drop onto the substrate can be reduced by decreasing the contact time of impacting drop, our findings may help to understand how endothermic animals lower hypothermia risks, and how insects stabilize their flight position during rainfalls. Here, we visualize such salient high-speed drop dynamics using a high-speed shadowgraphy and validate the distinctive features through experiments and theoretical models.

Droplet Impact on dry solid surfaces: Traditional vs Bio-mimetic , SAPTARSHI BASIL, DURBAR ROY, Department of Mechanical Engineering, Indian Institute of Science, KHUSHBOO PANDEY, Interdisciplinary Center for Energy Research (ICER), Indian Institute of Science, RABIRRATA MUKHERJEE, Department of Chemical Engineering, Indian Institute of Technology, Kharagpur — An experimental study of droplet impact has been conducted on four different substrates (2 traditional and 2 bio-mimetic) in the impact Weber number range of 6 to 130. The droplet shape dynamics have been visualized using high-speed shadowgraphy (at 10 kHz). Glass and PDMS are the traditional substrates, whereas the other two surfaces are inspired by rose-petal and lotus-leaf micro-structures. Various regimes are demarcated for all the substrates depending on the impact Weber number. The receding rebound, and breakup mechanisms of bio-mimetic surfaces are found to be strikingly different from that of the traditional substrates. Dimensional analysis, scaling arguments and energetics have been utilized to unearth the underlying dynamics of the impact.

Drop Impact on Liquid Film: Bouncing to Merging Transition for Two-Liquid System , ABHISHEK SAHA, XIAN WU, University of California San Diego — Impact of a drop on liquid film is critical in several industrial applications, including inkjet printing and thermal sprays. Previous studies using single liquid for both drop and film showed that the impact could result in two outcomes, namely bouncing and merging, and the transition between these two states is a function of impact Weber number and film thickness. It was also reported that for a range of Weber number, the impact outcome changes from merging to bouncing, back to merging and then to bouncing again, as we increase the film thickness. Since many of the advanced printing technologies such as 3D inkjet printers can print multiple materials, very often the drop and the impacted the liquid film are required to be of different liquids with varying properties. Recognizing its importance, in this talk, we will present a study on the dynamics of drop impact on liquid film using two liquids with similar surface tension, but different viscosities. The results with two-liquid systems show a shift in the transitional boundaries with respect to that of the single-liquid system. In addition to the two types of merging, early merging and late merging, reported for single-liquid systems, a third kind of merging was also observed for two-liquid systems. This third kind of merging was found to reduce the degree of non-monotonicity of the transitional boundaries between bouncing and merging states.
We have constructed a 26-meter-tall vacuum tube to study drop impacts at very high velocities, in a controlled manner. Under reduced ambient pressure we can attain impact velocities up to 23 m/s and Reynolds numbers as large as $10^5$. We focus on the ejecta sheets produced when the drop impacts on a thin layer of liquid. The ejecta dynamics are captured with two simultaneous high-speed video cameras, one focused on the ejecta details and the other on the overall crown evolution. Experiments are performed over a wide range of drop and liquid viscosity combinations. The ejecta breakup and splashing are greatly affected by the ambient air pressure, where air-drag and Bernoulli suction pressure can bend the sheets into intricate shapes. The tip of the sheet can bend up or down depending on the viscosity ratio between the two liquids. Under some impact conditions the sheet entraps a toroidal bubble as the crown rises away from the substrate. The sheets finally rupture to produce a spray with a myriad of micro-droplets. We construct a simplified model to describe the shape evolution.

---

**Monday, November 25, 2019 5:16PM - 7:13PM**

**Session P22 Drops: Multiple Drop Interactions**

604 - Alexander Wray, University of Strathclyde, Glasgow

**5:16PM P22.00001 Hydrodynamic Spin Lattices**

PEDRO SAENZ, UNC Chapel Hill, GIUSEPPE PUCCI, Institute of Physics of Rennes, SAM TURTON, ALEXIS GOUJON, RUBEN ROSALES, JORN DUNKEL, JOHN BUSH, MIT — In this talk, we will introduce a hydrodynamic analog system that allows us to investigate simultaneously the wave-mediated self-propulsion and interactions of effective spin degrees of freedom in inertial and rotating frames. Millimetric liquid droplets can walk across the surface of a vibrating fluid bath, self-propelled through a resonant interaction with their own guiding wave fields. A walking droplet, or ‘walker’, may be trapped by a submerged circular well at the bottom of the fluid bath, leading to a clockwise or counter-clockwise angular motion centered at the well. When a collection of such wells is arranged in a 1D or 2D lattice geometry, a thin fluid layer between wells enables wave-mediated interactions between neighboring walkers. Through experiments and mathematical modeling, we demonstrate the spontaneous emergence of coherent droplet rotation dynamics for different types of lattices. For sufficiently strong pair-coupling, wave interactions between neighboring droplets may induce local spin flips leading to ferromagnetic or antiferromagnetic order. Transitions between these two forms of order can be controlled by tuning the lattice parameters or by imposing a Coriolis force mimicking an external magnetic field.

**5:29PM P22.00002 The dipole approximation for droplets in Hele-Shaw flows**

YOAV GREEN, Ben-Gurion University of the Negev — For the past decade, the interaction between two droplets flowing in a Hele-Shaw cell has been modelled as a dipolar force. On one hand, the correspondence between experiments and the dipole model is remarkable. On the other hand, the dipole model doesn’t satisfy the required no-flux boundary condition at the droplet interfaces. Despite of this contradiction, the dipole model remained popular. In an attempt to bridge this gap, Sarig et al. [1] derived an exact analytical solution, for a two-droplet system satisfying zero-flux at both droplet interfaces, that showed remarkable correspondence to experiments. However, their solution was given in terms of an infinite Fourier series, which didn’t allow for the desired resolution of the contradiction. By deriving approximations for their Fourier series, for the cases of small and large droplet spacing, we resolved the contradiction [2]. We showed that the large spacing approximation reduces to the expected dipole-like solution. We extended our solution to arbitrary droplet numbers. Our infinite lattice solution includes a corrections term relative to the dipole model. Finally, our small spacing solution also provides a novel lower limit.


**5:42PM P22.00003 The Stability and Dynamics of Bouncing Droplet Rings**

MILES COUCHMAN, JOHN BUSH, Massachusetts Institute of Technology — Millimeter droplets bouncing on the surface of a vibrating fluid bath may interact through their shared wavefield to form bound states. In this talk, we present the results of a combined experimental and theoretical investigation of the stability of droplet rings. As the bath’s vibrational acceleration is increased progressively, droplet rings are observed to destabilize into a variety of dynamical states including steady rotational motion, radial oscillations, azimuthal oscillations, and azimuthally traveling waves. The instability observed is dependent on the ring’s initial radius and drop number, and whether the drop’s are bouncing in-or out-of-phase relative to their neighbors. As the vibrational acceleration is further increased, more exotic forms emerge including pentagonal and square structures. A linear stability analysis based on the trajectory equation of Couchman et al. (JFM, 2019) rationalizes the observed instabilities and provides insight into the dynamics of droplet lattices and other aggregates. Connections with vortex arrays in superfluid helium and Bose Einstein condensates are discussed.

[1] The authors gratefully acknowledge financial support from NSERC grant 502891 and NSF grants DMS-1614043 and CMMI-1727565.
its anticipated range of applicability, up to and including the limit of touching droplets. The shielding effect of other droplets is demonstrated, solutions of the full governing equations in order to verify the accuracy of the model; in particular, the model is found to perform well even outside evaporation of multiple thin sessile droplets under the assumption that the droplets are well separated. Comparisons are made with numerical

SON, ALEXANDER WRAY, BRIAN DUFFY, University of Strathclyde — An asymptotic model is derived for the competitive diffusion-limited evaporation of multiple thin sessile droplets under the assumption that the droplets are well separated. Comparisons are made with numerical

Relevant Impingement Conditions: a Computational Study

1

1National Science Foundation Graduate Research Fellowship Program

6:08PM P22.00005 Competitive evaporation of multiple sessile droplets, STEPHEN WILSON, ALEXANDER WRAY, BRIAN DUFFY, University of Strathclyde — An asymptotic model is derived for the competitive diffusion-limited evaporation of multiple thin sessile droplets under the assumption that the droplets are well separated. Comparisons are made with numerical solutions of the full governing equations in order to verify the accuracy of the model; in particular, the model is found to perform well even outside its anticipated range of applicability, up to and including the limit of touching droplets. The shielding effect of other droplets is demonstrated, and the model is used to investigate the role of this effect on shielding droplet evolutions and lifetimes, as well as on the coffee-ring effect.

6:21PM P22.00006 On Splashing Dynamics of Diesel Drop Trains Under Engine-Relevant Impingement Conditions: a Computational Study1, DAVID MARKT JR., ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth, ROBERTO TORELLI, Argonne National Laboratory, SEONG-YOUNG LEE, Michigan Technological University — In this study, advanced 3D simulations of micron-sized diesel drop trains impinging on a solid substrate are presented. The droplet size and impact velocity are representative of the impingement conditions during fuel injection in internal combustion engines. Generally, diesel fuel injection is studied using Lagrangian-Eulerian solvers which rely on spray-wall interaction sub-models to predict the drop impingement outcomes. We have found such sub-models may predict inaccurate splashing outcomes under such extreme impingement conditions. Therefore, using drop trains as an idealized spray, highly-resolved simulations are presented and quantities such as the splashed mass ratio and secondary droplet distribution are compared to common spray-wall interaction sub-models. The effects of liquid film thickness, contact angle and ambient gas pressure on the splashing dynamics are also quantified. By varying the impingement conditions, a robust analysis is presented highlighting the dominant parameters which affect splashing.

1This work is supported by the Department of Energy (EERE) and the Department of Defense (TARDEC), under Award Number DE-EE0007292. The funding provided by the Massachusetts Clean Energy Center is gratefully acknowledged.

6:34PM P22.00007 ABSTRACT WITHDRAWN –

6:47PM P22.00008 Manipulation of drop-drop interactions in millifluidic device using inlet spacing design, E. M. ARUN SANKAR, RAGHUNATHAN RENGASWAMY, Indian Institute of Technology Madras — Colloidal particles of different shapes and anisotropy are required for various applications. Researchers have reported synthesis of simple structures of colloidal particles in 1D microfluidic devices. Synthesis of more complicated 2D structures by contacting drops in a 2D device demands knowledge of their movement. When there are many drops in the system, each drop disturbs the flow field and interact with other drops. By appropriately manipulating these hydrodynamic interactions, one can arrange the drops in a desired shape. A simple model which captures the dominant interactions in 2D millichannel Hele-Shaw geometry, and can be used in an optimization framework, was proposed from our group. We noticed that, the hydrodynamic interactions between the drops and therefore the resulting droplet pattern formation are affected by the initial inter-droplet spacing. The spacing between the drops can be changed by injecting (withdrawing) the continuous phase fluid into the system. Here, we present manipulation of hydrodynamic interactions to arrange the drops in a desired shape in a millifluidic device by sending them into the channel at optimum spacing.

7:00PM P22.00009 Measurement of hydrodynamic forces on near-contact droplets1, REECE KEARNIE, GREGORY P BEWLEY, LINDA BU, Cornell University — Collisions between particles in turbulent flows are important in many natural and industrial processes, including rainfall and combustion. Though three-dimensional particle tracking is used routinely to measure the motions of fluid, detection of collisions between particles has been limited to non-automated methods due to the difficulty of differentiating between particles as they come close together. Near contact, particle pairs may experience forces that arise from interactions between the background flow and the motion of the two particles, and these forces may tend to promote or prevent collision. In our experiment, we observe water droplets ranging from 10 microns to 100s of microns in radius. We present measurements of the hydrodynamic interactions between droplets at separations of less than one diameter using an in-house particle tracking algorithm that resolves trajectories approaching and including contact.

1NSF grant 1605195

Monday, November 25, 2019 5:16PM - 7:13PM —

Session P23 Biological Fluid Dynamics : Cardiac Flows II

605 - Amir Arzani, Northern Arizona University
5:16PM P23.00001 In-vitro investigation of the effect of variable pulsatile flow and Left Ventricular Assist Device speed on the intraventricular hemodynamics. MARISSA MIRAMONTES, FANETTE CHASSAGNE, VENKAT KESHAV CHIVUKULA, JENNIFER BECKMAN, CLAUDIUS MAHR, ALBERTO ALISEDA, University of Washington, MECHANICAL AND CARDIOLOGY COLLABORATION — Left Ventricular Assist Devices (LVAD) are used to treat end-stage heart failure but induce unfavorable hemodynamics in the left ventricle (LV) that can result in thromboembolic and hemorrhagic events such as stroke. This study aims to quantify the impact of native contractility and LVAD speed on fluid mechanics inside the LV, and the associated thrombogenicity. Stereo Particle Image Velocimetry (PIV) measurements of the flow in a patient-specific LV flow phantom implanted with a real LVAD are analyzed under a wide range of clinically relevant parameters: pulsatility, preload and afterload. The combination of reduced pulsatility and LVAD speed results in highly heterogeneous flow patterns, with co-existing jet-like flow and high-residence time recirculating regions. Increased pulsatile flow and higher LVAD speeds improve velocity field variability, associated with less stagnation. Pulsatility plays a greater role in reducing stagnant regions compared to increasing LVAD speed. Unfavorable hemodynamics caused by decreased contractility combined with low LVAD speed may explain the persistent incidence of thrombosis even in new generation LVADs.

5:29PM P23.00002 Numerical Modeling of Thrombus Transport to the Cerebral Vasculature in the Presence of a Left Ventricular Assist Device. ANGELA STRACCIA, VENKAT KESHAV CHIVUKULA, FANETTE CHASSAGNE, JENNIFER BECKMAN, CLAUDIUS MAHR, ALBERTO ALISEDA, University of Washington — Left ventricular assist devices (LVAD) represent an increasingly available treatment for end-stage heart failure. Despite improvements in LVAD design that have greatly reduced the incidence of in-pump thrombosis, thromboembolic events, such as stroke, remain the main cause of mortality and morbidity. This study aims to identify the relationship between the source of thrombi – LVAD outflow graft or aortic valve – and their destination in the cerebral vasculature. The hemodynamics, from the aortic arch and LVAD outflow graft to the Circle of Willis, are investigated using 3D time-resolved computational fluid dynamics (CFD) in patient-specific models obtained from segmented medical imaging. Thrombi of different sizes are seeded in the blood flow and followed throughout their trajectories via Lagrangian particle tracking, accounting for inertial effects. We achieve a statistical description of the likelihood of thrombi being transported towards different regions of the cerebral vasculature by studying a wide distribution of thrombi properties and seeding locations. Patient-specific stroke risk, and its origins, is quantified, along with the altered hemodynamics in the cerebral vasculature in the presence of stroke.

5:42PM P23.00003 Further classification of the cardiac vortex: On the scaling of core vorticity with heart rate. GIUSEPPE DI LABBIO, LYES KADEM, Concordia University Montreal — The generation and persistence of a diastolic vortex ring within the healthy left ventricle has gained significant interest over the past two decades. This cardiac vortex has been shown to ease the transport of blood from inflow to outflow, preserving inflowing kinetic energy in a near-optimal manner while also promoting low blood residence time. Nonetheless, despite its practical interest and the multitude of associated diagnostic indices developed over the years, there has been no attempt at modeling the behavior of the cardiac vortex core. In this work, the flow in a healthy left ventricle was simulated in vitro in a double-activation left heart duplicator. The ensuing flow was captured using two-dimensional time-resolved particle image velocimetry in a clinically-relevant plane. Three heart rates were examined (50, 70 and 90 bpm). By comparison to the Lamb-Oseen-Hamel vortex, a one-parameter physics-based empirical fit is developed for the temporal evolution of the vorticity of the cardiac vortex core. The fitting constant is shown to be related to the circulation of the core by the end of the E wave of filling. With a suitable choice of scaling (dependent on the heart rate and ventricle width), the core vorticity at different heart rates appear to collapse onto a single curve.

5:55PM P23.00004 Computational study of blood flow patterns in a wall-deforming model of the left ventricle under healthy and LVAD-assisted conditions. TINGTING YANG, VENKAT KESHAV CHIVUKULA, ALBERTO ALISEDA, University of Washington, Seattle, WA, MULTIPHASE AND CARDIOVASCULAR FLOW LAB TEAM — Left ventricular assist device (LVAD) induce non-physiological flow patterns that are associated with thrombus formation. Numerical modeling of the hemodynamic environment inside simplified geometries of the human left ventricle incorporate wall deformation to simulate healthy and LVAD-assisted conditions. Input boundary conditions and wall deformation are derived from patient-specific measurements. Q-criterion, lambda-2 and streamline visualizations are used to compare the healthy and LVAD cases, better characterizing the impact of LVAD implantation on intraventricular flow patterns. Two vortex rings are formed during mitral filling that break down as they flow towards the ventricular apex, with the remaining filaments washed out through the aortic valve or inflow cannula. Small scale vortices, unreported in previous research with rigid ventricle walls, are found between the apex and the outer cannula wall. A methodology to investigate the physics underlying ventricular filling and emptying with different levels of ventricular contractility (ejection fraction) is demonstrated. LVAD surgical configurations, such as the inflow cannula and outflow graft angles, are studied to provide insights on risk assessment of these surgical techniques.

6:08PM P23.00005 Simulating Cardiac Fluid-Structure Interaction by an Immersed Finite Element Method. BOYCE GRIFFITH, MARSHALL DAVEY, University of North Carolina at Chapel Hill, CHARLES PUELZ, New York University, SIMONE ROSSI, MARGARET ANNE SMITH, DAVID WELLS, University of North Carolina at Chapel Hill — Cardiovascular diseases remain the leading causes of death worldwide, and tools for improving diagnosis, treatment planning, or medical device design promise to have a major impact on patient health. Simulating cardiac fluid dynamics across the cardiac cycle motivates the development of models that account for interactions between the blood, the muscular walls of the heart, and the thin cardiac valves. The immersed boundary method is a numerical approach to such problems of fluid-structure interaction (FSI). This talk will describe ongoing work to develop medical image-based models of cardiac FSI for modeling cardiac fluid dynamics. Our model of the heart includes image-based descriptions of the major anatomical features of the heart, including the atria and ventricles, and the nearby great vessels along with idealized anatomical models of the cardiac valves and experimentally constrained biomechanical models. To simulate cardiac FSI, we use an immersed boundary method that employs a finite element description of the immersed structure that enables the use of large-deformation nonlinear elasticity. The talk will outline the numerical approach and model formulation, and will investigate the role of pericardial tethering on establishing realistic intracardiac fluid flows.

1We gratefully acknowledge NSF awards OAC 1450327 and OAC 1652541 and NIH awards HL117063 and HL143336.
6:21PM P23.00006 Implantation orientation effects of a bileaflet mechanical heart valve in an anatomic left ventricle-aorta configuration1, HOSEIN ASADI, MOHAMMADALI HEDAYAT, IMAN BORAZJANI, Texas A&M University — Three-dimensional high-resolution simulations of a bileaflet mechanical heart valve (BMHV) have been carried out for an anatomic left ventricle-aorta configuration. The geometry of the anatomic left ventricle (LV) is reconstructed from MRI scanned images of a healthy subject and its motion is prescribed based on a lumped parameter model. The highly validated multi-block sharp interface curvilinear immersed boundary method (CURVIB) fluid-structure interaction (FSI) solver is used in which anatomic aorta and LV are discretized with a boundary-conforming and non-conforming curvilinear meshes, respectively. The motion of BMHV is calculated using the strong coupled FSI accelerated with Aitken convergence technique. The simulations are performed for three valve orientations, differing 45° from each other for two cardiac cycles. The kinematics of heart valves and instantaneous hemodynamics of each case, as well as, shear stress and platelet activation are analyzed to investigate the performance of each orientation.

1This work was partially supported by National Science Foundation award CBET 1453982 and the American Heart Association Grant 13SDG17220022. The computational resources were provided by the High-Performance Research Computing (HPRC) center at Texas A&M University.

6:34PM P23.00007 An Immersed Interface Method for Biomedical Fluid Structure Interaction, EBRAHIM KOLAHDOUZ, University of North Carolina at Chapel Hill; BRENT CRAVEN, United States Food and Drug Administration, BOYCE GRIFFITH, University of North Carolina at Chapel Hill — The mechanical interaction of incompressible viscous flows with immersed bodies is ubiquitously found in medicine and biology. A fluid structure interaction (FSI) coupling strategy is presented within the framework of the immersed interface method that allows fluid and solid subproblems to be solved in a partitioned manner and coupled through interface conditions. The present FSI approach allows for general complex geometries with discrete surfaces while retaining sharp resolution of stresses at the fluid-solid interface. In the coupling of the fluid to the solid, the interfacial fluid stresses drive the solid motion, and a penalty method is used to ensure that the fluid satisfies the no-slip condition along the fluid-solid interface. This approach enables the use of unstructured finite element discretizations of the solid domain while making use of structured-grid solvers for the incompressible Navier-Stokes equations. The algorithm is systematically verified and validated through comparisons with numerical and experimental benchmarks of increasing complexity. Applications of this method to biomedical applications, including the dynamics of bileaflet mechanical heart valves and dynamics of deformable blood clots inside IVC filter will be presented.

6:47PM P23.00008 Optimal bending rigidity of the aortic valve leaflets, YE CHEN, HAOXIAO CHEN, IANG LUO, Vanderbilt University — Proper bending stiffness and the ability to quickly respond to the dynamic pressure load on their surfaces are critical for the heart valves to carry out their physiological functions. Studies on how the flexibility of the leaflets affects the fluid-structure interaction (FSI) of heart valves are still very limited. In this talk, three-dimensional FSI simulations of a bioprosthetic aortic valve are performed using a parallelized immersed-boundary method. The pressure distribution over the leaflets and transient force on the valve are calculated. The thickness of the leaflets is varied from 0.1 mm to 0.8 mm, which results in a wide range of non-dimensional bending rigidity $EB^*$, normalized by the transvalvar pressure gradient. For valves with low bending rigidity ($EB^* \approx 1.0 \times 10^{-2}$), the valve functions normally and produces physiological characteristics of healthy valves. However, exceedingly low rigidity (for example, $EB^* \approx 1.2 \times 10^{-3}$) leads to flapping motion of the leaflets and impairs the valve’s performance. Stiffer valves ($EB^* > 0.2$) are more difficult to open and slower to close, which leads to higher resistance and a reduced flow rate during systole. The results reveal the existence of an optimal range of bending rigidity for the valve, where $EB^*$ is roughly between 0.003 and 0.04. Effects of bending rigidity on valve deformation and flow characteristics will also be discussed.

7:00PM P23.00009 Aortic Wall Shear Analysis from Asymmetric Prosthetic Heart Valve Design, ALEXANDROS ROSAKIS, MORTEZA GHARIB, California Institute of Technology — Asymmetrically stiffened aortic trileaflet valve leaflets can divert the systolic flow away from the stiffened leaflets and into the wall of the aortic root. Asymmetrical stiffening of the aortic valve can be caused by aortic valve stenosis. We show the different flow profiles caused by different combinations of stiffened and unstiffened leaflets. Furthermore, we describe how this asymmetric stiffening alters the wall shear stress on a model aorta. This investigation can be used to better understand disease states in patients with aortic valve stenosis. Moreover, understanding the effects of asymmetric leaflet stiffening on the aortic flow may allow us to design patient specific prosthetic polymer heart valves. These valves would have carefully tuned leaflet thicknesses that would direct the systolic flow in a way that minimizes damage to the patient’s aortic wall and better resembles natural healthy heart valves.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P24 Bubbly Flow II 606 - Jack Keeler, University of Manchester

5:16PM P24.00001 Experimental and numerical study of bubble transport within multiphase cross flow over a cylinder, ERIC THACHER, SIMO MAKIHARJU, University of California, Berkeley — Vortex-induced vibration from cross flow over a cylinder is an important design consideration in numerous applications. For single phase flow, this phenomenon has been studied extensively; however, while past researchers have shown that increasing phase fraction decreases vibration amplitude while increasing shedding frequency, the mechanisms causing these changes are not fully understood. Studying individual bubble transport may provide insight, as indicated by Voutsinas et al. (2009) who demonstrated that the frequency shift depends on bubble size. In this work we begin by studying the flow of individual bubbles over a cylinder in cross-flow, to assess the time needed for bubbles to be captured in the shed vortices. Following the method of Oweis et al. (2005), the capture time is predicted using a point-particle tracking model, as a function of bubble size, release position, and flow rate. The numerical results are then verified experimentally using high speed camera visualization of a cylinder in cross flow within a vertical flow loop. The time-resolved transport of a single stream of monodisperse bubbles from a needle and co-flow apparatus is used to assess the impact of capture time on cylinder pressure fluctuations, before expanding the study to higher void fraction flows. -/abstract- Authors: Eric W. Thacher and Simo A. M
5:29 PM P24.00002 Modeling and validation of bubble-induced fluctuation in bubbly flows.\(^1\) JUBEOM LEE, HYUNGMIN PARK, Seoul National University — The nature of two-phase turbulence is of a great interest academically and practically. In general, it is decomposed into two contributions: one from the shear-induced turbulence in the absence of bubbles and the other from the bubble-induced turbulence (agitation or fluctuation). Regarding the latter, it is further broken down into non-turbulent (a potential drift) and turbulent (perturbations due to bubble wake) parts. In the present work, we are interested in this bubble-induced fluctuation and suggest a model, which is derived based on a mixing length model (analogous to the single-phase flow turbulence) but considering the effect of neighboring bubble and bubble-induced liquid flow. As a result, we suggest models for turbulent and streamwise normal stress that includes the contributions from gradients of liquid velocity, void distribution, and bubble velocity. The novelty of this work is, we try to combine the non-turbulent and turbulent parts together. Finally, we validate our models with available data in the literature, including our own, obtained from various flow configurations such as a bubble-swarm, laminar pipe, and bubbly wake flows. We discuss the advantages and limitations of our models.

\(^1\)Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

5:42 PM P24.00003 Some notes on eddy viscosity in wall-bounded turbulent bubbly flows.\(^1\) TIAN MA, Duke University, Department of Civil & Environmental Engineering, YIXIANG LIAO, DIRK LUCAS, Helmholtz-Zentrum Dresden-Rossendorf, Institute of Fluid Dynamics, ANDREW BRAGG, Duke University, Department of Civil & Environmental Engineering, HELMHOLZ-ZENTRUM DRESDEN-ROSSENDORF, INSTITUTE OF FLUID DYNAMICS TEAM, DUKE UNIVERSITY, DEPARTMENT OF CIVIL & ENVIRONMENTAL ENGINEERING TEAM — Recently, based on data from DNS, Ma et al. (Phys. Rev. Fluids 2, 034301, 2017) proposed a model for closing the bubble-induced turbulence (BIT) in a typical Euler-Euler two-equation model, which appears to yield improved performance for predicting $k$ and $\epsilon$ over the previous models. The present study departs from this BIT model and purpose to use the same DNS data to investigate the behavior of the $C_\mu$ constant and standard eddy viscosity definition. It can be shown that $C_\mu$ constant computed using the DNS database has a very different behavior than that in single-phase flow. Checking closely, the deficiency originates from the description of the standard eddy viscosity that is intrinsic to this general hierarchy of Euler-Euler $k-\epsilon$ type model, hence, cannot be overcome by a more complex correction function for $C_\mu$. Departing from this point, a modification to the definition of the eddy viscosity in bubbly flows is derived for the Euler-Euler two-equation models. We focus on the intermediate region—a region extended from the core region, where bubble-induced production and dissipation are nearly in balance, and find that the modified model can lead to significantly improved predictions for the mean liquid, when compared with DNS data.

5:55 PM P24.00004 ABSTRACT WITHDRAWN —

6:08 PM P24.00005 Numerical two-phase flows study in channels with variable cross-section.\(^1\) GUSTAVO R. ANJOS, COPPE/Federal University of Rio de Janeiro — Bubbles and drops dynamics through capillaries of variable cross-section still remains of considerable importance in two-phase flows. The aim of this work is to investigate two-phase flows found in the cooling of new generation of computer processor units in the scope of the ThermoSMART - Marie-Curie/RISE Consortium. We seek to study numerically the effects of domain boundaries to the bubble dynamics, including change in the film thickness, bubble shape and vortex shedding in channels with variable cross-section using a moving mesh/boundary domain scheme, which dramatically shortens the domain length. Such a scheme moves the computational boundary nodes according to the bubble’s center of mass relative to the variable cross-section of a given problem. The new methodology proposed to simulate two-phase flows in variable cross-sectioned channels shows good accuracy to describe interface forces and bubble dynamics in different complex geometries with moving boundaries.

\(^1\)ThermoSMART (EU FP7-PEOPLE-2011-IRSES 294905), FAPERJ - Brazil

6:21 PM P24.00006 Bubbly flow in upward 90-degree elbow pipe: bubble dispersion and liquid flow structure\(^1\) HONGSEOK Choi, HYUNGMIN PARK, Seoul National University — In gas-liquid 2-phase flows, interfacial structure plays an important role in determining the transport characteristics, which changes with geometry and inlet condition. We experimentally investigate change in bubble dynamics and flow structures for gas-liquid bubbly flow in 90-degree bent square pipe, varying mean void fraction up to 3.0%. Continuous phase flows are chosen as laminar (Re = 550) and turbulent flows (Re = 7,000). We acquire the liquid-phase velocity using two-phase PIV technique, while gas-phase velocity and size distribution are measured with high-speed shadowgraphy. In laminar flow, bubbles move much faster than the liquid phase, resulting in a backflow and large recirculation region at the inner wall of the pipe. The size of this region increases with mean void fraction, inducing strong turbulence at the boundary. For turbulent flow, flow structure doesn’t show significant change with considered void fraction, but bubble trajectories move from the inner wall to the outer wall as mean void fraction increases. As a result, location of the maximum liquid-phase turbulence changes accordingly. Analysis of the interfacial force balance and mechanism for flow structure change will be discussed further.

\(^1\)Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

6:34 PM P24.00007 Transient bubble dynamics in a constricted Hele-Shaw channel.\(^1\) ANTOINE GAILLARD, JACK KEELER, MCND (University of Manchester), GREGOIRE LE LAY, Ecole normale superieure de Paris , GREGOIRE LEMOULT, ANNE JUEL, ALICE THOMPSON, ANDREW HAZEL, MCND (University of Manchester) — We explore the applicability of dynamical systems concepts recently used to study the transition to turbulence in shear flows to other subcritical transitions in fluid mechanics. We are ultimately interested in the subcritical instability of the linearly stable Saffman-Taylor finger in a Hele-Shaw channel, where finite perturbations can initiate complex dynamics for sufficiently large values of the driving parameter. Here, we concentrate on a geometrically- perturbed Hele-Shaw channel which supports multiple stable modes. We experimentally investigate and classify the different time-evolution scenarios of an air bubble of given volume when varying both the flow rate and the initial bubble shape. As the flow rate increases, the bubble exhibits increasingly complex behaviors, including oscillatory deformations and transient explorations of multiple-tipped unstable modes which often lead to bubble breakup, followed by multiple bubble interactions. Besides, we show that long and disordered transients can be observed at large flow rates depending on the level of noise in the system.

\(^1\)Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.
In this talk, we attempt to understand the transition to disorder in this system for a bubble in Hele-Shaw channel by finding invariant solutions, in the form of steady states and periodic orbits, of the governing equations. Inspired by recent developments in the transition to turbulence in shear flow, and using dynamical systems theory, we discuss the idea, that when the flow-rate is large enough the bubble will transiently explore the stable manifolds of weakly unstable edge states of the system.

**Monday, November 25, 2019 5:16PM - 7:13PM – Session P25 Bubbles: Acoustics**

**5:16PM P25.00001 Model for lipid-encapsulated microbubbles using transient network theory**, BASHIR M. ALNAJAR, University of Colorado Colorado Springs, SHANKAR L. SRIDHAR, MARK A. BORDEN, FRANCK J. VERNEREY, University of Colorado Boulder, MICHAEL L. CALVISI, University of Colorado Colorado Springs — Encapsulated microbubbles (EMBs) are widely used to enhance contrast in ultrasound sonography and are finding increasing use in biomedical therapies such as drug/gene delivery and tissue ablation. EMBs consist of a gas core surrounded by a stabilizing shell made of various materials, including polymers, lipids, and proteins. Lipid-coated EMBs present a unique challenge for modeling due to their relatively large oscillations and nonlinear, viscoelastic properties. We propose a novel model for a lipid-coated, spherical EMB that utilizes a statistically-based continuum theory based on transient networks to simulate the encapsulating material. The use of transient network theory permits the viscoelastic properties of the encapsulation — such as stress, elastic energy and entropy — to be calculated locally based on the configuration of lipid molecules. The model requires a minimum number of parameters that include the lipid concentration, and the rates of attachment and detachment of lipids to and from the network. The model closely reproducibly the experimentally-measured radial response of an ultrasonically-driven, lipid-coated microbubble. The model also reproduces experimentally-observed nonlinear behavior, such as compression and expansion-dominated oscillations.

**5:29PM P25.00002 Direct Numerical Simulation of weakly nonlinear bubble oscillations**, YUZHE FAN, HAISEN LI, Acoustic Science and Technology Laboratory, Harbin Engineering University, Harbin 150001, China, DANIEL FUSTER, Sorbonne Université, Centre National de la Recherche Scientifique, UMR 7190, Institut Jean Le Rond D’Alembert, F-75005 Paris, France — Bubble dynamics have been studied extensively during the last century. Because of the highly nonlinear nature, the oscillations of gas bubbles in a liquid are of great practical interest and relevant in diverse technologies. Traditionally, to study stably oscillating micro-bubbles subjected to ultrasound or megasonic waves, Rayleigh-Plesset (R-P) type equations assuming spherical symmetry are frequently chosen. However, although the bubble can oscillate without breaking in such weakly nonlinear oscillation regimes, non-spherical effects neglected in R-P models will become increasingly important with the increasing of the amplitude of pressure pulse and the frequency of the incident wave. To investigate such non-spherical mechanisms, we present simulations of the interaction between a traveling acoustic wave and a single bubble using Direct Numerical Simulations (DNS) [Fuster and Popinet. (2018). J. Comput. Phys. 374, 752-768]. Depending on the pressure waves emitted by the moving non-spherical bubble, we characterize different weakly nonlinear oscillation regimes as a function of the Reynolds, Weber and the amplitude of incident pressure. The comparison between DNS and R-P type equations is also done to quantitatively estimate the range of applicability of simplified models.

**5:42PM P25.00003 Multiscale modeling of bubble acoustics**, SUHAS JAIN, JAVIER URZAY, ALI MANI, PARVIZ MOIN, Center for Turbulence Research — The detection of bubbles and the accurate determination of their sizes is important for several naval applications, including the characterization of ships and submarine wakes. One practical way of accomplishing this task is by using acoustic methods, in which the bubbles are sized using the dependence of their resonance frequency on their equilibrium radius. This work presents an algorithm that accurately captures the interface, maintains favorable characteristics such as boundedness and total-variation diminishing properties for the volume fraction, and is fully non-dissipative and numerically stable. In practical oceanic flows, there is a large separation of scales between the sizes of the bubbles and the correlation wavelengths of the free surface, and therefore their acoustic scattering behaviors are also different. Taking advantage of this disparity of scales and the resulting scattering behavior, a modeling technique for bubble-size detection is explored where the bubbles are modeled as point scatterers, whereas the free surface is resolved by the diffuse-interface method.

First author acknowledges the support from the Franklin P. and Caroline M. Johnson Fellowship.
The acoustically-driven expansion and collapse of a near wall bubble. A high-order accurate, fully compressible, multiphase model is used to simulate the acoustically-driven expansion and collapse of a gas bubble in water. The Rayleigh collapse has been the standard model of collapse: it uses initially static distributions of pressures for which the pressures in the two domains are unequal and uniform. The model presented here initiates the bubble dynamics and the surrounding fluid dynamics by an oscillating moving boundary condition. The results of the acoustically induced collapse are compared to those of the Rayleigh growth and collapse. A small standoff distance greatly restricts the sphericity during the bubble's expansion stage. For similar maximum bubble volumes, the acoustical-driven collapse predicts much higher wall pressures than the Rayleigh model.

Single Bubble Collapse at Audible Frequencies and High Amplitudes

While the characteristics of bubbles' radial motion have been widely studied for driving frequencies higher than 20 kHz, lower frequency ranges remain unexplored. In this work, dynamics of an acoustically forced gas/vapor single micro-bubble in water have been studied for driving frequencies below 20 kHz by means of a reduced-order model accounting for all the critical thermo-mechanical contributions. Our investigations in a large parameter space (frequency x amplitude = [1-100 kHz] x [1-7.5 atm]) suggest that at low frequencies and/or high amplitudes, water phase changes play a major role in determining the bubble dynamics, yielding a remarkably different behavior where the first bubble collapse is not necessarily the strongest one (which is the case for higher frequencies). However, at moderate amplitudes (1-1.1 atm), low frequency forcing yields bubble dynamics comparable to the high-frequency/high-amplitude cases.

This work was supported by the Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC), an NWO Gravitation programme funded by the Ministry of Education, Culture and Science of the government of the Netherlands.

Ultrasound contrast agents are micron-sized bubbles used to increase contrast in ultrasound imaging. These microbubbles can be steered in the systemic circulation by the ultrasound-induced Bjerknes force, presenting a potential for highly-directed thrombolysis or cancer chemotherapy. While the dynamics of a single microbubble under ultrasound excitation are well understood, the dynamics of a suspension of microbubbles in physiologically-realistic flows are understudied: a greater understanding of the competition between hydrodynamic and ultrasound-induced forces in this flow regime would enable clinical applications. Experiments conducted in a cylindrical tube at physiologically relevant Reynolds numbers and under various pressure amplitudes and pulse repetition frequencies (PRF) will be presented. In-house high-speed tracking characterized the forces acting on the microbubbles by calculating microbubble velocities and accelerations. When the Bjerknes force is small, the microbubbles are only affected by the shear-induced lift force in the flow. At higher pressures and PRFs, the Bjerknes force overcomes the shear-induced lift force and displaces the microbubbles in the direction of ultrasound propagation, potentially putting them in contact with the arterial wall for drug delivery.

Ultrasound-enhanced mass transfer during single bubble diffusive growth

This project is co-funded by ThermaSMART under the European Unions Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No. 778104.

Ultrasound contrast agents are micron-sized bubbles used to increase contrast in ultrasound imaging. These microbubbles can be steered in the systemic circulation by the ultrasound-induced Bjerknes force, presenting a potential for highly-directed thrombolysis or cancer chemotherapy. While the dynamics of a single microbubble under ultrasound excitation are well understood, the dynamics of a suspension of microbubbles in physiologically-realistic flows are understudied: a greater understanding of the competition between hydrodynamic and ultrasound-induced forces in this flow regime would enable clinical applications. Experiments conducted in a cylindrical tube at physiologically relevant Reynolds numbers and under various pressure amplitudes and pulse repetition frequencies (PRF) will be presented. In-house high-speed tracking characterized the forces acting on the microbubbles by calculating microbubble velocities and accelerations. When the Bjerknes force is small, the microbubbles are only affected by the shear-induced lift force in the flow. At higher pressures and PRFs, the Bjerknes force overcomes the shear-induced lift force and displaces the microbubbles in the direction of ultrasound propagation, potentially putting them in contact with the arterial wall for drug delivery.

Ultrasound contrast agents are micron-sized bubbles used to increase contrast in ultrasound imaging. These microbubbles can be steered in the systemic circulation by the ultrasound-induced Bjerknes force, presenting a potential for highly-directed thrombolysis or cancer chemotherapy. While the dynamics of a single microbubble under ultrasound excitation are well understood, the dynamics of a suspension of microbubbles in physiologically-realistic flows are understudied: a greater understanding of the competition between hydrodynamic and ultrasound-induced forces in this flow regime would enable clinical applications. Experiments conducted in a cylindrical tube at physiologically relevant Reynolds numbers and under various pressure amplitudes and pulse repetition frequencies (PRF) will be presented. In-house high-speed tracking characterized the forces acting on the microbubbles by calculating microbubble velocities and accelerations. When the Bjerknes force is small, the microbubbles are only affected by the shear-induced lift force in the flow. At higher pressures and PRFs, the Bjerknes force overcomes the shear-induced lift force and displaces the microbubbles in the direction of ultrasound propagation, potentially putting them in contact with the arterial wall for drug delivery.

This project is co-funded by ThermaSMART under the European Unions Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No. 778104.

Ultrasound contrast agents are micron-sized bubbles used to increase contrast in ultrasound imaging. These microbubbles can be steered in the systemic circulation by the ultrasound-induced Bjerknes force, presenting a potential for highly-directed thrombolysis or cancer chemotherapy. While the dynamics of a single microbubble under ultrasound excitation are well understood, the dynamics of a suspension of microbubbles in physiologically-realistic flows are understudied: a greater understanding of the competition between hydrodynamic and ultrasound-induced forces in this flow regime would enable clinical applications. Experiments conducted in a cylindrical tube at physiologically relevant Reynolds numbers and under various pressure amplitudes and pulse repetition frequencies (PRF) will be presented. In-house high-speed tracking characterized the forces acting on the microbubbles by calculating microbubble velocities and accelerations. When the Bjerknes force is small, the microbubbles are only affected by the shear-induced lift force in the flow. At higher pressures and PRFs, the Bjerknes force overcomes the shear-induced lift force and displaces the microbubbles in the direction of ultrasound propagation, potentially putting them in contact with the arterial wall for drug delivery.

Ultrasound contrast agents are micron-sized bubbles used to increase contrast in ultrasound imaging. These microbubbles can be steered in the systemic circulation by the ultrasound-induced Bjerknes force, presenting a potential for highly-directed thrombolysis or cancer chemotherapy. While the dynamics of a single microbubble under ultrasound excitation are well understood, the dynamics of a suspension of microbubbles in physiologically-realistic flows are understudied: a greater understanding of the competition between hydrodynamic and ultrasound-induced forces in this flow regime would enable clinical applications. Experiments conducted in a cylindrical tube at physiologically relevant Reynolds numbers and under various pressure amplitudes and pulse repetition frequencies (PRF) will be presented. In-house high-speed tracking characterized the forces acting on the microbubbles by calculating microbubble velocities and accelerations. When the Bjerknes force is small, the microbubbles are only affected by the shear-induced lift force in the flow. At higher pressures and PRFs, the Bjerknes force overcomes the shear-induced lift force and displaces the microbubbles in the direction of ultrasound propagation, potentially putting them in contact with the arterial wall for drug delivery.
5:16PM P26.00001 The Unsteady Suction Actuator1, NIMROD SHAY, BAR MIZRAHI, OFEK DRORI, ARIEL YANIV, AVRAHAM SEIFERT, Tel Aviv University — Steady suction is known to be significantly more effective than steady blowing for boundary layer separation control. Pulsed blowing is known to be significantly more effective than steady blowing, owing to the instability and favorable timescales imposed by unsteady 3D vorticity components created by a suitable device. Recently it was shown that unsteady suction can be more effective even when compared to steady suction. This paper describes the development and characterization of a new fluidic device, capable of creating unsteady suction with no moving parts. The device uses at least one fluidic oscillator and at least one ejector to create unsteady suction. Detailed numerical and experimental rapid prototyping techniques were used and the results of the device characterization in still fluid are described. It is shown that the new device can create the required output signals for external flow separation control using relatively low input mass flux and low power. Further to its development, it was already tested interacting with a turbulent boundary layer and compared directly to other flow control devices in keeping an airfoil boundary layer attached. [1] Seifert, A. and Pack, L.G. “Active flow separation control on wall-mounted hump at high Reynolds numbers,” AIAA J., vol. 40, pp. 1363–1372, 2012. [2] Morgulis, N. and Seifert, A., “Fluidic flow control for improved performance of small Darrieus type wind turbines,” Wind Energy, Volume 19, Issue 9, September 2016, Pages 1585-1602.

5:29PM P26.00002 On the Interaction of Unsteady Suction and Turbulent Flat Plate Boundary Layer, BAR MIZRAHI, AVRAHAM SEIFERT, Tel Aviv University — This paper describes the interaction of unsteady suction created by the new oscillatory suction (OSUB) actuator with a turbulent, flat plate boundary layer. Unsteady suction is imposed on the boundary layer from four discrete holes in the plate in the transitional region of the boundary layer. The boundary layer is first characterized without excitation then two magnitudes and two frequencies for each magnitude are imposed. The resulting flow is very effectively modified by the imposed suction. Hot-wire scans of planes parallel to the wall at two different heights and full boundary layer velocity profiles along selected rays from the origin of the disturbances reveal a fascinating 3D complex yet quite coherent patterns. Phase-locked analysis decomposes the controlled flow field and reveals the unsteady flow created by the unsteady suction. The disturbances are compared to those created by discrete steady suction through the same array of four spanwise holes and reveal the parameter range in which unsteady suction has significant benefits as compared to steady suction.

5:42PM P26.00003 A Comparison Between Oscillatory and Steady Suction for Airfoil Separation Control1, AVRAHAM SEIFERT, BAR MIZRAHI, Tel Aviv University — This paper presents a direct comparison between two classes of flow control actuators. The first actuator uses steady suction while the other actuator uses unsteady suction. Both use also pulsed blowing (PB) downstream of the suction locations. Both actuators were installed on a custom-designed and 3D printed airfoil, specifically designed to have two distinct separation locations. At the upstream location, four suction holes each are placed symmetrically to the chord and are connected either to steady or unsteady suction devices. Further downstream, where the boundary layer tends to separate again, a pair of PB slots are placed, connected each pair to each device, again symmetrically to the chord line. The airfoil is mounted on a load cell and aerodynamic forces are measured at low speeds. The actuators were previously calibrated on a benchtop setup. Tests are conducted first in the absence of actuation, at low Reynolds numbers (up to 200K), then the two actuation concepts are directly compared. The range of parameters in which unsteady suction has benefits over steady suction for separation control are identified. Actuation energy expenditure is shown to be significantly lower when using unsteady suction.

5:55PM P26.00004 Preventing boundary layer separation by non-uniform suction control1, JAMES RAMSAW, MATHIEU SELLIER, University of Canterbury, WEI HUA HO, University of South Africa — Suction of the boundary layer has been studied as a method of flow control for over a hundred years. However, its use has failed to migrate to mainstream engineering applications. There are two main reasons for this: one, the relative complexity and broad parameter space for the control makes it difficult and costly to design; two, the energy required to generate the suction can often outweigh the savings from reduced drag or improved performance. To address these issues, an investigation of suction control on laminar flow around the circular cylinder and within an axisymmetric diverging pipe has been performed. Numerical simulations were performed with optimisation to determine the most effective and efficient applications of suction to achieve a variety of objectives. The models without suction were validated against experimental data from the literature. With the aim of making the design of suction control more accessible in real applications, the focus of the results was on the relationships between the uncontrolled flow and the optimised control parameters. Particular attention was paid to the objective of controlling the separation of the boundary layer and its potential use as an objective, or as an important feature, in determining the best control.

6:08PM P26.00005 Control of Shock-Induced Separation and Vorticity Concentrations in a Serpentine Diffuser1, TRAVIS BURROWS2, BOJAN VUKASINOVIC3, ARI GLEZER4, Georgia Institute of Technology — Advanced propulsion inlet systems utilize complex serpentine diffusers, whose geometry engenders large-scale streamwise vortices and boundary-layer separation coupled to shock formation at high flow rates that limit engine efficiency operating range due to severe losses and distortion. The present experimental investigation demonstrates active control of the diffuser transonic shock by utilization of surface-mounted fluidic oscillating jets, thereby indirectly control streamwise vorticity concentrations by exploiting the coupling to the shock. It is demonstrated that flow control modifies the shock topology and footprint, confining it towards the diffuser’s sidewalls. Consequently, the streamwise vortices are displaced, ultimately mitigating their advection of low-momentum fluid into core diffuser flow. This active flow control leads to a 35% reduction in average diffuser circumferential distortion and a concomitant increase in pressure recovery, indicating the capability of this flow control approach to extend the engine operation range beyond its nominal operating condition.

1Supported by ONR
2Graduate Research Assistant, Woodruff School of Mechanical Engineering
3Research Engineer, Woodruff School of Mechanical Engineering
4Professor, Woodruff School of Mechanical Engineering
6:21PM P26.00006 Flow Physics and Scaling for Discrete Jet Forcing on a Wall-Mounted Hump. CHRISTOPHER OTTO, BENJAMIN CHAMPION, JESSE LITTLE, The University of Arizona, RENE WOSZIDLO, The Boeing Company — An experimental study is conducted to explore flow physics and scaling parameters (e.g., aspect ratio, exit area, spacing) for various types of fluidic oscillators in support of the development of active flow control technology. Various actuation modules are designed, built and tested on an existing model of the NASA hump geometry. Experiments are carried out at a Reynolds numbers of 1.0 x 10^6 (Ma = 0.09). Time-averaged pressure measurements are conducted along both the chord and span of the model. Stereoscopic PIV is performed downstream of the actuation location to investigate the underlying control mechanisms in detail. Flow control using various spatially distributed fluidic oscillators was applied for spacings of Δz/c = 4.55% & 9.09%. Performance curves were compared based on a momentum, mass flow, and particle image velocimetry. Of specific interest are the roles of independently-interchangeable variations between the inlet flow (M = 0.8) and crosswind speed (up 35 knots) on the on-set and evolution of topology of separation. It is shown that for a given inlet flow the presence of sufficiently high cross winds lead to the formation of a three-dimensional separation domain that has an azimuthal, horseshoe-like boundary with its tip near the windward edge. As the cross wind speed increases, separation spreads by the formation of secondary interacting azimuthal separation cells whose topology resembles the main separation domain. Fluidic control of the separation leads to significant reduction in losses that are manifested by reduction in cross stream total pressure deficit and a concomitant increase in the inlets mass flow rate.

1Supported by the Boeing Company.

6:34PM P26.00007 Fluidic Control of Round Inlet Flow in Cross Wind1. D.A. NICHOLS, B. VUKASINOVIC, A. GLEZER, Georgia Institute of Technology, M. DEFORE, B. RAFFERTY, The Boeing Company — The suction flow within a round inlet in the presence of cross wind is investigated experimentally with specific emphasis on characterization and control of separation over the surface of windward lip using arrays of surface static pressure ports and radial total pressure rakes, surface oil-flow visualization, and particle image velocimetry. The nominally 2-D curved surface is designed to promote separation representative of that seen in rotorcraft applications. A spanwise array of fluidic oscillating jets located upstream of the separation provides dissipative, high-frequency actuation, which in turn controls the characteristic scale of the separation domain. The effect of the actuation and its progression from the upstream to the downstream edges of the separated bubble are assessed using stereoscopic particle image velocimetry. Emphasis is placed on local entrainment and flow features near the actuators and their ensuing downstream progression throughout the separated region. Additionally, the relationship between the local flow features at separation and characteristics of the ensuing separated flow will be assessed with respect to the global flow control effectiveness.

1Supported by VLRCOE.

6:47PM P26.00008 Flow Topology and Control of a Closed Separation Domain Over a Curved Surface1. C.J. PETERSON, N.K. KOUKPAIZAN, B. VUKASINOVIC, M.J. SMITH, A. GLEZER, Georgia Institute of Technology — The unsteady interactions between fluidic oscillating jets and the vorticity concentration within a separation bubble formed by a subsonic cross flow over a curved surface is investigated experimentally. The nominally 2-D curved surface is designed to promote separation representative of that seen in rotorcraft applications. A spanwise array of fluidic oscillating jets located upstream of the separation provides dissipative, high-frequency actuation, which in turn controls the characteristic scale of the separation domain. The effect of the actuation and its progression from the upstream to the downstream edges of the separated bubble are assessed using stereoscopic particle image velocimetry. Emphasis is placed on local entrainment and flow features near the actuators and their ensuing downstream progression throughout the separated region. Additionally, the relationship between the local flow features at separation and characteristics of the ensuing separated flow will be assessed with respect to the global flow control effectiveness.

1Supported by ARO.

7:00PM P26.00009 Controlled Aerodynamic Forces on Slender Axisymmetric Bodies at High Incidence1. E. LEE, Y. HUANG, B. VUKASINOVIC, A. GLEZER, Georgia Institute of Technology — The flow and aerodynamic loads over slender axisymmetric bodies at high incidence are dominated by interactions of a hierarchy of vortical structures resulting from streamwise- successive, wake instabilities of the cylinders forebody, main, and aft segments. These vortical interactions are driven by the onset of a counter-rotating vortex pair that form over the forebody and are extremely receptive to small perturbations and can rapidly evolve asymmetrically resulting in significant side force and yawing moment. The present wind tunnel investigations utilize an axisymmetric model (L/D = 10, incidence up to 65 o , Re D = 810 4 ) to exploit the receptivity of the forebody flow to small perturbations for controlling the evolution of the forebody vortices and thereby the aerodynamic loads (side forces and roll and yaw moments). Upwind actuation is effected using synthetic jet actuators at the juncture of several forebodies of different aspect ratios. It is shown that the flow is extremely receptive to actuation at high incidence angles yielding side force changes of up to C S = 5 (corresponding C L about 6.5). The effected side forces are used to control the models lateral stability.

1Supported by ARO.

Monday, November 25, 2019 5:16PM - 7:00PM – Session P27 Biological Fluid Dynamics: High Re Swimming II 609 - Jennifer Franck, U Wisconsin

5:16PM P27.00001 Combined Heaving and Pitching Propulsors of Non-Uniform Flexibility1. AMIN MIVECHI, TIANJUN HAN, Lehigh University, MEHDI SAADAT, GEORGE V. LAUDER, Harvard University, KEITH W. MOORED, Lehigh University — Many swimming animals propel themselves efficiently through water by oscillating their fins with combined heaving and pitching motions. Contrary to most research on flexible propulsors, these fins are not homogenously flexible, but rather their flexibility varies along their chord and span. Here, using a simple flexibility model we experimentally examine the effect of the distribution of flexibility on the performance of a three-dimensional propulsor using combined heaving and pitching motions. We manufacture propulsors with a combination of rigid and flexible materials where their proportion compared to the chord length is determined by the flexion ratio. The experiments are conducted in a recirculating water channel by varying the frequency of motion at Re = 10^4, with the pitching motion lagging the heaving motion by 90 degrees. Both the effective flexibility and the flexion ratio of the propulsors are varied independently and force, power and amplitude data are recorded for each experiment. We find peak efficiencies greater than 60% and we detail the dependency of the performance on the variation in the structural properties of the propulsor.

1Office of Naval Research under Program Director Dr R. Brizzolara on MURI grant number N00014-08-1-0642.
5:29PM P27.00002 Tailoring the Bending Pattern of Non-Uniformly Flexible Pitching Propulsors Enhances Propulsive Efficiency1, JIANJUN HAN, MELEKE KURT, AMIN MIVEHCHI, KEITH W. MOORED, Lehigh University — Aquatic animals swim rapidly and efficiently by using flexible propulsors. It has been observed that bio-propulsors typically have non-uniform flexibility with increasing flexibility towards the trailing edge. However, most previous research has examined the effect of uniform flexibility on bio-propulsion. Here, we conduct experiments on three-dimensional propulsors with non-uniform chordwise flexibility. We used a simple step function distribution where there is a rigid leading-edge section and a finite flexibility trailing-edge section. This piecewise distribution is defined by the flexion ratio, that is, the ratio of the rigid section length to the chord length. The forces and moments are measured for purely pitching propulsors with various flexion ratios and effective flexibilities. Peak efficiencies for these purely pitching propulsors reach as high as 56% and occur above the first resonant frequency of the system. Additionally, a scaling relation for the resonant frequency of non-uniformly flexible propulsors is determined. This work highlights the result that not only is the resonant condition important for high efficiency performance, but also the bending pattern of a propulsor.

1Supported by the Office of Naval Research under Program Director Dr. Bob Brizolara, MURI grant number N00014-14-1-0533.

5:42PM P27.00003 Efficient optimization of swimming gaits1, DANIEL FLORYAN, XUANHONG AN, CLARENCE W. ROWLEY, Princeton University — We study a simplified model of fish swimming—namely a rigid foil undergoing periodic motion—seeking motions that are optimal in regards to a particular objective (e.g. maximal thrust production). We use an immersed boundary method, and develop an adjoint formulation that allows us to efficiently calculate the gradient of the objective function that is used with gradient-based optimization. Moreover, the adjoint field provides sensitivity information which can be used to elucidate the physics responsible for optimality.

1Supported under ONR MURI Grant N00014-14-1-0533, Program Manager Robert Brizzolara.

5:55PM P27.00004 Hydrodynamic performance of a self-propelled ray with oscillatory locomotion1, YOUNG DAL JEONG, JAE HWA LEE, UNIST — Mylobatoid rays have dorsoventrally flattened diamond-shaped bodies with expanded pectoral fins, and they swim by oscillating their large pectoral fins. This oscillatory locomotion utilizes the lift-based propulsion, which is specialized for efficient propulsion. Inspired by the high efficient rays, we perform numerical simulations of a self-propelled oscillating ray in a viscous quiescent flow. To consider the fluid-structure interaction between the oscillating ray and surrounding fluid, the penalty immersed boundary method is adopted. From references for the biological ray kinematics, the oscillatory locomotion in the vertical and spanwise directions is actively given at the leading edge, although the chordwise moving motion is freely movable by the fluid-flexible body interaction. Moreover, three-dimensional motion of the rest part is passively determined, and the propulsion performance is influenced by the elastic properties. The effects of the elastic properties on the cruising speed and the input power are investigated systematically.

1This research was supported by the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2017R1D1A1A09000537) and the Ministry of Science, ICT & Future Planning (NRF-2017R1A5A1015311).

6:08PM P27.00005 Intermittent locomotion of a self-propelled plate.1, JAEHA RYU, HYUNG JIN SUNG, Department of Mechanical Engineering, KAIST — Many fish and marina animals swim by using a combination of the active bursting phase and the passive coasting phase, which is known as the burst-and-coast swimming. The immersed boundary method is applied to explore the intermittent locomotion of a three-dimensional self-propelled flexible plate. The degree of the intermittent locomotion is captured in a duty cycle (DC = \( T_b/T_f \), which is the ratio of the interval of the burst phase \( T_b \) to the total flapping period \( T_f \)) of the active flapping motion. The averaged cruising speed \( (U_c) \), the input power \( (P) \), and the swimming efficiency \( (\eta) \) are analyzed as a function of the duty cycle \( (DC) \). The maximum \( U_c \) is obtained at \( DC = 0.9 \), while the maximum \( \eta \) is at \( DC = 0.3 \). The hydrodynamics by the intermittent locomotion is scrutinized by using the superimposed flag and the phase map. The characteristics of the flapping motion are demonstrated at the bursting and coasting phases, respectively. The modal analysis is performed to examine how the flapping motion plays a role in the propulsion mechanism. The velocity map and the vortical structures are visualized to show the influence of the intermittent locomotion qualitatively and quantitatively.

1This work was supported by a grant from the National Research Foundation of Korea (NRF) (No. 2019M3C1B7025091).

6:21PM P27.00006 CFD-Based Multi-Objective Controller Optimization for Soft Robotic Fish with Muscle-like Actuation, ANDREW HESS, XIAOBO TAN, TONG GAO, Michigan State University — Soft robots take advantage of rich nonlinear dynamics and large degrees of freedom to perform actions by novel means beyond the capability of conventional rigid robots. Nevertheless, there have been considerable challenges in analysis, design, and optimization of soft robots due to their complex behaviors. This is especially true for underwater soft robotic swimmers whose dynamics are determined by highly nonlinear fluid-structure interactions. We present a holistic computation framework that employs a multi-objective evolutionary method to optimize feedback-based controllers of a soft robotic fish prototype subject to artificial muscle actuation. The resultant nonlinear fluid-structure interactions are fully solved by using novel fictitious domain active strain method which specify the entire-body curvature variation, we demonstrate that imposing contractile active strains locally can produce various swimming gaits using far fewer control parameters. It also facilitates feedback controller design for muscle actuation schemes using high-fidelity CFD simulation data. We optimize the controller coefficients via several “thought” experiments where we seek optimal swimming performances of the robotic fish tracking moving targets.

6:34PM P27.00007 Scaling Laws for Three-Dimensional Combined Heaving and Pitching Propulsors1, FATMA AYANCIK, KEITH MOORED, Lehigh University — The underlying physics of oscillatory swimming can be captured with simple models based on the scaling of the added mass and circulatory forces. Here, by considering both of these forces, we present new scaling relations for three-dimensional combined heaving and pitching propulsors with varying aspect ratios. Classic linear theory is augmented by additional nonlinearities and modified for three-dimensional effects by considering the added mass of a finite-span propulsor, the downwash/upwash effects from the trailing vortex system and the elliptical topology of shedding trailing-edge vortices. We verified the scaling relations by using experiments and self-propelled inviscid numerical simulations over a wide range of variables including the dimensionless amplitude, dimensionless heave-to-pitch ratio, Strouhal number, and aspect ratio. The developed relations are found to be in excellent agreement with the numerical and experimental data. These scaling laws are used to identify physical mechanisms that influence thrust and efficiency, and as a guide for improving performance.

1Supported by the Office of Naval Research under Program Director Dr. Bob Brizolara, MURI grant number N00014-14-1-0533.
6:47PM P27.00008 Geometric Influence on Force and Frequency Response of a Bioinspired Undulated Cylinder \quad KATHLEEN M. LYONS, University of Wisconsin - Madison, WI \quad CHRISTIN T. MURPHY, ANDREW GIARENDI, Naval Undersea Warfare Center - Newport, RI \quad JENNIFER A. FRANCK, University of Wisconsin - Madison, WI — The unique three-dimensional geometry of seal whiskers has been shown to significantly change the turbulent structures directly downstream, resulting in reduced forces and dampened vibrations when compared with circular cylinders. Biomimicry gives an opportunity to apply the unique whisker geometry to other areas of engineering that require vibration suppression, frequency tuning, or force reduction. This computational investigation isolates the complex geometric parameters of the seal whisker via an undulated cylinder model prescribed by seven non-dimensional parameters including undulation wavelength, thickness, slenderness, amplitudes in the streamwise and transverse flow directions, as well as a peak-shift and a symmetry parameter that induce a non-sinusoidal periodic undulation. Using a two-factor fractional factorial design of experiments, these simulations demonstrate the geometric parameters and two-parameter interactions that are most influential for reducing drag, root-mean-square lift force, and shifting the frequency spectra. It is shown that both transverse and streamwise undulation amplitudes have a notable impact on the downstream vortex structures, thus impacting the frequency spectra and recirculation region via different mechanisms.


5:16PM P28.00001 A simple hydride model for cerium ejecta particles\textsuperscript{1} \quad JONATHAN D. REGELE, JOHN D. SCHWARZKOPF, WILLIAM T. BÜTTLER, ALAN K. HARRISON, Los Alamos National Laboratory — Cerium ejecta particles created by shock driven Richtmyer-Meshkov instabilities are known to hydrde inside of deuterium gas and release exothermic energy in the form of heat and increased particle temperature. Cerium dihydride, which is the reaction product specie, is solid under the experimental conditions considered. A model is developed to describe the hydriding process by combining the lumped-capacitance thermal conduction model, Ranz-Marshall heat transfer correlation, and a diffusion-controlled reaction. Comparisons with experimental data are used to determine model accuracy and to discover what additional physics should be considered in the model.

\textsuperscript{1}Support from DOE ASC PEM Mix and Burn is gratefully acknowledged.

5:29PM P28.00002 Modeling pseudo-turbulence in compressible particle-laden flows\textsuperscript{1} \quad GREGORY SHALLCROSS, University of Michigan, RODNEY FOX, Iowa State University, JESSE CAPECELATRO, University of Michigan — When a shock passes through a dense suspension of solid particles, velocity fluctuations are generated in particle interstitial sites. While this is captured in fully resolved simulations of shock-particle interactions, it remains a challenge to reproduce using coarse-grained models, such as Eulerian-Eulerian and Eulerian-Lagrangian methods. Recent work has revealed that pseudo-turbulent kinetic energy (PTKE) can contribute significantly to the overall kinetic energy during shock-particle interactions. We demonstrate this term acts to systematically increase the local Mach number, and needs to be accounted for to properly capture particle dispersion. A transport equation for PTKE is presented, and closure for the dissipation rate is proposed. The equations are implementing in a high-order Eulerian-Lagrangian framework and compared against direct numerical simulations of shock-particle interactions. We demonstrate the model is capable of predicting the pseudo-turbulent Reynolds stresses with correct levels of anisotropy, independent of the drag law employed. Finally, a stochastic model informed by the PTKE is proposed to improve the prediction of particle dispersion.

\textsuperscript{1}This work was supported by a NASA Space Technology Research Fellowship

5:42PM P28.00003 Examination of Particle Force Model and its Uncertainty in a Detonation-Driven Multiphase Flow\textsuperscript{1} \quad JOSHUA GARNO, FREDERICK OUELLET, RAHUL KONERU, THOMAS JACKSON, S. BALACHANDAR, University of Florida, BERTRAND ROLLIN, Embry-Riddle Aeronautical University — Recent work in the compressible, multiphase flow community has shown that the compressible Maxey-Riley-Gatignol (MRG) force model captures the transient forces exerted on a particle by a passing air shock and compressed flow. In this work, the model’s predictive capability in the post-detonation flow regime is considered following a rigorous study of the explosive products as the carrier phase. The model parameters of the JWL equation of state are varied to observe their individual sensitivities on the post-detonation flow. Uncertainty quantification with experimental data provides the most influential and likely JWL parameters. With the gas phase in agreement with experiments, the validity of the compressible MRG model under extreme condition is reviewed. Experimental X-ray data provides the trajectory of a few Tungsten particles ejected from an initial explosion. Particle trajectory data from experiments is compared with finite-volume, Euler-Lagrange, point-particle simulations results employing the MRG model.

\textsuperscript{1}This work was supported (in part) by the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, under Contract No. DE-NA0002378.

5:55PM P28.00004 Analyzing particle curtains with advection-corrected correlation image velocimetry and particle image velocimetry\textsuperscript{1} \quad JANGHAN PARK, DANIEL FREELONG, PATRICK WAYNE, PETER VOROBIEFF, University of New Mexico — We conduct an experimental study of an interaction between a planar shock in air and a nominally planar curtain of particles embedded in air. We investigate shocks at Mach numbers 1.2, 1.4, 1.7, and 2.0. Particle curtains of different nominal thickness (2, 4, and 6 mm) are subjected to shock acceleration. The particles are soda lime microspheres with a density of 1.4 g/cc and diameters ranging from 30 to 50 microns. The curtain formation prior to shock arrival is recorded by a high-speed camera during 2 seconds at 960 frames per second. The curtain mass flow rate is also acquired. Each frame of the particle curtain video is analyzed with advection-corrected correlation image velocimetry (ACCIV) and particle image velocimetry (PIV). We investigate whether ACCIV offers any advantages over PIV for this flow. Measurements of velocity combined with mass flow rate data can then be used to provide an estimate of the particle volume fraction and insights into the air entrainment by the curtain. The subsequent study must relate the local volume fraction in the curtain and its thickness with the post-shock features we observe.

\textsuperscript{1}This research is partially supported by NSF grant 1603915.
6:08PM P28.00005 Shock interaction with particle curtains of varying thickness. DANIEL FREELONG, PATRICK WAYNE, JANGHAN PARK, GREGORY VIGIL, CAROLINA SHAHEEN, Student Member, PETER VOROBL-EFF, Professional Member — The interaction of a curtain of particles with a shock wave is investigated experimentally. Soda lime particles form a gravity-driven curtain. The geometry of the curtain-forming nozzle can be adjusted, producing curtains with a nominal thickness of 2 mm, 4 mm, and 6 mm in the direction of the shock. Particle volume fractions for all three curtains range between 1% and 9%, with variations primarily due to particle acceleration along the vertical extent of the curtain. Prior to shock impact, we measure the instantaneous and average velocities of the particles to show that the particles are nearly in a free-falling state, with their average velocity increasing linearly with vertical distance. Experimental data reveal that the shock wave is both transmitted through and partially reflected by the curtain. Time-resolved images show the underlying flow structure of the interaction. This research is supported by the US Defense Threat Reduction Agency (DTRA) grant HDTRA1-18-1-002.

6:21PM P28.00006 Mach number and particle size effects on the unsteady drag of shocked micro-droplets. KYLE HUGHES, ADAM MARTINEZ, JOHN CHARONKO, Los Alamos National Laboratory — Experiments of shock-accelerated micro-droplets show high drag coefficients when the particles are tracked from initial acceleration through the relaxation times. An eight-pulse particle tracking diagnostic measures individual particle positions, and a shadowgraph system measures shock location, with pressure transducers providing shock speed at the test section. These diagnostics give us detailed measurements of particle positions versus time for Mach 1.2, 1.3 and 1.4 experiments, allowing us to calculate accelerations and drag. Comparison is made to previous experiments conducted on solid Nylon particles in similar flow regimes.

6:34PM P28.00007 Evaluation of Point-Particle Models in Shock-Particle Bed Interactions. RAHUL BABU KONERU, University of Florida, BERTRAND ROLLIN, Embry-Riddle Aeronautical University, FREDER-ICK OUELLET, S. BALACHANDAR, University of Florida — In this work, 3D Euler-Lagrange (EL) point-particle simulations of shock-particle cloud interaction are presented for two cases (i) shock interacting with a stationary bed of particles and (ii) a multiphase shock tube. In an effort to improve the point-particle models, results from these EL simulations are compared against particle-resolved (PR) Euler simulations in case of the stationary bed and experiments from the Multiphase Shock Tube facility at Sandia National Laboratories (SNL). In the stationary bed simulations, it is observed that at low incident shock Mach numbers and particle volume fractions (10%-15%), the point-particle models predict the average gas properties reasonably well. As the effects of compressibility become more prominent (presence of bow shocks), the models predict a higher drag than that is observed in the PR simulations. A sensitivity analysis is performed to identify the force components responsible for this additional drag. In the case of the multiphase shock tube, the effects of particle collisions and the initial curtain profile on the curtain expansion rate are explored. The particle collisions in this case are modeled using a soft-sphere type DEM model in CMT-Nek.

6:47PM P28.00008 Implementation of Key Capabilities to Study Unsteady Drag of Shock-Accelerated Particles with an Arbitrary Lagrangian-Eulerian Code. TANNER NIELSEN, Los Alamos National Laboratory, W. CURTIS MAXON, University of Missouri, NICK DENISSEN, Los Alamos National Laboratory — The dynamic drag coefficient on particles due to shock-acceleration has been observed, experimentally and computationally, to significantly increase during the passage of the shock over the particle. The later times, after the passage of the shock during which the particle is accelerated to the post-shock conditions, have not been as thoroughly studied nor are there models that accurately capture drag effects in this unsteady regime. This work details key capabilities that have recently been added to FLAG, an arbitrary Lagrangian-Eulerian (ALE) code developed at Los Alamos National Laboratory, to enable the study of shock-accelerated particles. This unique simulation tool allows high-resolution studies of a single particle from rest to post-shock acceleration. The ALE framework permits the particle to move freely within the computational domain based on the pressure and viscous forces. Details will be given regarding the implementation of the viscous terms in FLAG to enable the solution of the Navier-Stokes equations for the air surrounding the particle. The drag calculation is done in a way that allows the integration of the forces on the particle as it moves freely through the domain.

7:00PM P28.00009 High Resolution Simulations of Particle Acceleration in Shock-Driven Multiphase Flows. WILLIAM MAXON, Self, TANNER NIELSEN, Collaborator, NICHOLAS DENISSEN, JONATHAN REGELE, Mentor, JACOB MCFARLAND, Advisor, UNIVERSITY OF MISSOURI-COLUMBIA TEAM, LOS ALAMOS NATIONAL LABORA-TORY TEAM — Particle drag models, which capture macro viscous and pressure effects, have been developed over the years for various flow regimes to enable cost effective simulations of particle-laden flows. The relatively recent derivation by Maxey and Riley has provided an exact equation of motion for spherical particles in a flow field based on the continuum assumption. Many models that have been simplified from these equations have provided reasonable approximations; however, the sensitivity of the shock-driven multiphase instability to particle drag requires a very accurate model to simulate. To develop such a model, 2D axisymmetric and 3D Cartesian Navier-Stokes DNS of a single particle in a transient, shock-driven flow field were conducted in the hydrocode FLAG. FLAG’s capability to run arbitrary Lagrangian-Eulerian (ALE) hydrodynamics coupled with solid mechanic models in solids makes it an ideal code to capture the physics of the flow field around the particle as it is shock-accelerated — a challenging regime to study. Preliminary results have shown higher drag than the current models predict. Simulation results will be used to create a new drag model for multiphase particle-in-cell methods.

Monday, November 25, 2019 5:16PM - 7:00PM – Session P29 Biological Fluid Dynamics : Red Blood Cells and Hemodynamics
5:16PM P29.00001 Dynamics of RBCs under shear flow in sickle cell disease. A tool for monitoring the clinical state of patients.1 EMANUELLE HELFER, MAXIME SAHUN, SCOTT ATWELL, ALEXANDER HORNUNG, ANNE CHARRIER, ANNIE VIALLAT, Aix Marseille Univ, CNRS CINaM, CATHERINE BADENS, Aix Marseille Univ, INSERM, MMG — The regimes of motion of red blood cells (RBCs) under shear flow have been extensively studied because they directly relate to the cell mechanical properties. In addition to tumbling and tanktreading motions, other regimes were recently discovered, such as swinging, flip-flopping, and rolling. Computational studies provide complex phase diagrams of motion that depend on the ratio of RBC cytoplasm to external fluid viscosities and on the capillary number. Surprisingly, no experiments have been performed on RBCs from patients with sickle cell disease (SCD) which have altered mechanical properties. Here, we show that the dynamics of SCD-RBCs is modified and correlates with change in RBC density and state of hydration. Though the tumbling - flip-flopping - rolling path observed by increasing shear rate is not changed for SCD, the rolling to tanktreading threshold occurs at higher shear stress. We use this feature to propose a mechanical marker, namely the fraction of tanktreading RBCs in a large cell population, to follow the clinical state of SCD patients and to predict the very handicapping vaso-occlusive crises. We show that this marker is patient-dependent and is stable over time in the “out of crisis” period while it strongly varies during the course of a crisis.

1This work has been carried out thanks to the support of A*MIDEX.

5:29PM P29.00002 Flow-induced Segregation and Dynamics of Red Blood Cells in Sickle Cell Disease1 XIAO ZHANG, University of Wisconsin-Madison, CHRISTINA CARUSO, WILBUR A. LAM, Emory University, MICHAEL D. GRAHAM, University of Wisconsin-Madison — Blood flow in sickle cell disease (SCD) can substantially differ from the normal due to significant alterations in the physical properties of sickle red blood cells (RBCs). Chronic complications, such as endothelial dysfunction, are associated with SCD, for reasons that are unclear. Direct numerical simulations are performed to investigate the dynamics of a binary suspension of flexible biconcave discoids and stiff curved prolate spheroids that represent healthy and sickle RBCs, respectively, in plane Poiseuille flow. The key observation is that the sickles exhibit a strong margination towards the walls. The marginated sickle RBCs roll like rigid bodies and may poke and damage the walls due to their stiffness and “spikiness”. A simplified drift-diffusion model, which incorporates hydrodynamic migration and pair collisions, predicts a greater margination in a binary suspension containing a small fraction of stiff cells than in a pure suspension of stiff cells, which may explain the experimental observation that a heterogeneous blood sample containing healthy (flexible) RBCs with a small fraction of stiffened RBCs causes more severe endothelial inflammation and dysfunction than a homogeneous sample of stiffened RBCs.

1This work is supported by NIH grant 1R21MD011590-01A1.

5:42PM P29.00003 3 D Classification of Red Blood Cells in microchannels. CHRISTIAN WAGNER, Saarland University — Red blood cells (RBCs) are very soft objects that can pass capillaries smaller than the cell's diameter. Due to their high deformability, they couple strongly with the flow and can adopt many different shapes. For their quantitative characterization we developed a new confocal 3D imaging technique for fluorescent stained RBCs. We found two equilibrium cell shapes under certain flow condition: the so called ‘slipper’ and the ‘croissant’ shape. Numerical simulations are in good agreement with experimental observations. In addition, high throughput data of classical 2-D microscopy combined with an adaptive neural network allow us to obtain the full phase diagram of RBC shape as a function of the flow rate. In larger channels, we use the confocal technique to characterize the margination of single rigidified RBCs in a suspension of healthy RBCs. Margination of e.g. white blood cells or platelets at the vessel walls is a haemodynamic key mechanism of our immune system. Our confocal observation technique allows us to characterize the distribution of hard vs. soft cells in full time and space resolution for the first time. Again numerical simulations are in good agreement although some quantitative differences remain that need further investigations.

5:55PM P29.00004 Reduced-order Models for Migration and Shear-induced Diffusion of Red Blood Cells in Simple Geometries1. HARRY WANG, JOSEPH SHERWOOD, OMAR MATAR, Imperial College London — From a fluid dynamics perspective, blood can be treated as a suspension of highly-deformable red blood cells (RBCs). The RBCs migrate away from the walls of the confining vessel primarily due to their deformability, resulting in inhomogeneous distributions, distinctive in the microvasculature. Migration away from the walls is countered by shear-induced diffusion effects due to hydrodynamic particle-particle interactions. While mesoscale simulations can capture RBC dynamics well, there is a need for more efficient continuum models that can accurately model RBCs distributions in larger networks that describe the microvasculature. Here, we study the behaviour of RBC suspensions in simple flow configurations using reduced-order models. We use a drift-diffusion equation to describe the evolution of the RBC concentration, coupled to balance equations for the bulk mixture. Different forms for migration and shear-induced diffusion terms in the drift-diffusion equation are compared in rectangular and cylindrical geometries. The problem is reduced to two-dimensions using appropriate scaling, which exploits geometrical length-scale disparity, and asymptotic reduction. Velocity and concentration profiles predicted by our simulations are compared to experimental data in the literature.

1PhD funding from EPSRC CDT Fluid Dynamics across Scales for HW is acknowledged.

6:08PM P29.00005 Red cell-resolved blood flow modeling in in vivo-like microvascular networks: predicting hemodynamic changes due to loss of red cell deformability1. SAMIAN EBRAHIMI, PROSENJIT BAGCHI, Rutgers University — Microvascular networks in human body are made of the smallest blood vessels, and responsible for gas and nutrient transport to tissues, and regulation of blood flow in individual organs. The architecture of a microvascular network is complex and characterized by bifurcating, merging and tortuous vessels. Blood in such small vessels behaves as a concentrated suspension primarily made of red blood cells (RBC) which are extremely deformable. We developed a 3D simulation technique to model flow of deformable RBCs in physiologically realistic microvascular networks that are comprised of multiple bifurcating and merging vessels. The model is versatile, and can consider networks irrespective of topological/geometrical complexities. It provides fully 3D and detailed information of hemodynamic quantities, such as RBC partitioning at bifurcations, cell-free layer, and wall shear stress. Many diseases, such as sickle cell disease, malaria and diabetes mellitus, are associated with a loss of RBC deformability. A detailed quantification of changes in microvascular hemodynamics under such conditions is lacking. Using the model, we provide the first-ever simulation results on the changes in network-scale blood flow under varying RBC deformability. The specific focus is on retinal microcirculation which is known to be adversely affected due to loss of RBC deformability.

1Funded by National Science Foundation.
force required for the synthetic tissue to rupture through the cerclage stitch is recorded. The results of this study provide insight into the most
suture material used in the cerclage are varied. The synthetic cervices are stitched by physicians according to clinical techniques. The maximum
silicone to mimic physiological, softening cervical tissue. Aspects of the cervical geometry (length of cervix, shape and width of dilatation) and
physicians from The George Washington University Hospital, we create generalized, synthetic models of the cervix and fabricate them with
in which the uterine cervix softens, shortens and dilates before reaching full term, usually between 18 and 22 weeks gestation, such that a
1
bifurcation was almost independent of the flow rate through the channel, the velocity of the lateral shift was suggested to be nearly proportional
and exhibited a lateral shift from the outer wall to the inner wall until NWE was developed. Since the development length of NWE from the
bifurcation was almost independent of the flow rate through the channel, the velocity of the lateral shift was suggested to be nearly proportional
to the main flow velocity.

1JSPS KAKENHI 17H03176

Monday, November 25, 2019 5:16PM - 7:13PM –
Session P30 Biological Fluid Dynamics : Medical Devices 612 - Wesley Harris, MIT

5:16PM P30.00001 The impact of cervical geometry and suture material on cerclage integrity . ALEXIA BAUMER, The George Washington University, ALEXIS GIMOVSKY, The George Washington University Medical Faculty Associates, MEGAN C LEFTWICH, The George Washington University — Cervical insufficiency is a medical condition during pregnancy in which the uterine cervix softens, shortens and dilates before reaching full term, usually between 18 and 22 weeks gestation, such that a preterm birth occurs. It is a common cause of second trimester pregnancy loss. Part of the clinical treatment of this condition is the cervical cerclage, a procedure to close the cervix with a purse-string stitch. There are conflicting findings on the efficacy of the cerclage, with most studies relying on statistical evidence. The purpose of this investigation is to examine the mechanical limitations of the cerclage. Working with physicians from The George Washington University Hospital, we create generalized, synthetic models of the cervix and fabricate them with silicone to mimic physiological, softening cervical tissue. Aspects of the cervical geometry (length of cervix, shape and width of dilatation) and suture material used in the cerclage are varied. The synthetic cervices are stitched by physicians according to clinical techniques. The maximum force required for the synthetic tissue to rupture through the cerclage stitch is recorded. The results of this study provide insight into the most effective clinical interventions and the mechanism of their success.

5:29PM P30.00002 A Magnetorheological Hemostatic Agent . YONATAN TEKLEAB, Massachusetts Institute of Technology, NIKOLAOS KOKOROSKOS, GEORGE VELMAHOS, Massachusetts General Hospital, GARETH MCKINLEY, WESLEY HARRIS, Massachusetts Institute of Technology — Magnetorheological (MR) suspensions are used in systems requiring responsive fluids with fast-acting, tunable properties. MR valves, have been effective in rapidly and locally arresting pressure-driven flows in mechanical systems. Motivated by these applications, we developed a magnetorheologically-actuated valve for use in the human body, to reduce or halt hemorrhage. 80% of trauma related deaths in the first hour of hospital admission are due to hemorrhagic shock. Such a hemostat would provide a prehospital intervention opportunity to stem bleeding, giving physicians more time to resuscitate patients upon trauma facility admission. The valve comprises an injectable, biocompatible MR suspension with externally-placed permanent magnets. To trigger the MR effect near the injury site in bleeding patients, the fluid was designed for biocompatibility, rapid injectability, and local actuation within blood vessels. We have synthesized, characterized, and demonstrated efficacy, through benchtop and in vivo rat tests, a novel, minimally invasive, MR hemostatic agent. Using magnets in 3D printed holders that can be worn by field surgeons, we demonstrate arrest of a major hemorrhagic event, dramatically reduced lost blood volume, sustained blood pressure, and significantly increased survival time.

5:42PM P30.00003 Influence on the fluid mechanics of blood flow of heterogeneity and anisotropy of the coil mass deployed inside intracranial aneurysms . JULIA ROMERO BHATHAL, Univ. Grenoble Alpes, CNRS, Grenoble INP, 3SR, Grenoble, France, FANETTE CHASSAGNE, LAUREL MARSH, Dpt. of Mechanical Engineering, Univ. of Washington, Seattle, WA, MIKE LEVITT, Dpt. of Neurological Surgery, Univ. of Washington, Seattle, WA, USA, CHRISTIAN GEINDREAU, Univ. Grenoble Alpes, CNRS, Grenoble INP, 3SR, Grenoble, France, ALBERTO ALISEDA, Dpt. of Mechanical Engineering, Univ. of Washington, Seattle, WA — Cerebral aneurysms are often treated with coils to induce thrombosis. Flow in coiled aneurysms has been studied extensively through CFD simulations to predict the flow after endovascular treatment. Representing coil mass as a homogeneous porous medium represents a limitation to the understanding of flow in coiled aneurysms and inhibits clinically accurate predictions. We present the characterization of the spatial heterogeneity and anisotropy of the coil mass deployed inside the aneurysmal sac based on synchrotron X-ray tomography of aneurysm phantoms treated with real coils and describe the resulting flow based on a parametrization of coil’s permeability that combines a theoretical model with the experimental results. The segmented images are used to compute the mean porosity, the porosity gradient along the inertial axis of the coil, the porosity map and the corresponding permeability map for different types of spatial discretization. The results show that the porosity varies between 0.6 to 0.95, leading to a factor 100 in permeability variations. CFD simulations results compare our results with standard homogeneous isotropic modeling.
5:55PM P30.00004 Effect of varying inhalation durations in normal breathing and HFOV conditions. MANIKANTAM GADDAM, YU FENG, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Experimental and computational studies in idealized and subject-specific airways at normal breathing conditions (Womersley number, $Wo=2-4$) and in high-frequency oscillatory ventilation (HFOV, $Wo=4-25$) have shown flow separation at bifurcations, secondary flows, and steady streaming at the end of inhalation. However, the effects of varying inhalation duration (relative to breathing time period) on the flow field remain unclear. We conducted 2D simulations on a Weibel airway model representing mouth to trachea with G2 to investigate the influence of inhalation time (IT) to breathing time (BT) in normal breathing and HFOV conditions. Oscillatory breathing patterns, with peak inhalation at Reynolds number ($Re$) of 1070 and peak exhalation at $Re$ of 1100, were prescribed as inflow conditions for different $Wo$ values. With increasing $Wo$ for a given IT/BT ratio, residual flow region increased at the end of exhalation, affecting flow during inhalation in the next breathing cycle. With increasing IT/BT, residual flow region increased at the end of exhalation for Wo ranging from 2.4 to 7.5 in the oral region.

This work was supported by a Carroll M. Leonard Faculty Fellowship to Santhanakrishnan.

6:08PM P30.00005 Steady streaming and conditional turbulence in high-frequency ventilation. JUSTIN LEONTINI, CHINTHAKA JACOB, Swinburne University of Technology, DAVID TINGAY, Murdoch Children’s Research Institute — High-frequency ventilation (HFV) is a technique used to ventilate neonates and patients with critical respiratory distress syndrome. It uses very fast yet shallow inflations, resulting in small peak pressures, thereby protecting lungs from over-distension. There are several mechanisms proposed for the gas transport during HFV, and here we investigate two of the primary ones; turbulent mixing and mean streaming. We have conducted direct numerical simulations of these high-frequency reciprocating flows in 1:2 bifurcations with geometric proportions relevant to the first five generations of the neonatal airway. Conditional turbulence is observed in the first three generations of the airway, the turbulence occurring when the flow rate is near its maximum. The results suggest this turbulence is generated via an instability of the Dean vortices generated via the curvature of the bifurcation. This mechanism differs from that which leads to conditional turbulence in the reciprocating flow in a straight pipe. We also quantify the recirculating flow rate due to mean streaming and find it to be around 5% of the maximum flow rate in the upper airway - this is enough to provide adequate gas exchange during HFV.

1CJ acknowledges the financial support of Swinburne University of Technology and the Murdoch Children's Research Institute via a growth SUPRA. This work was performed on the OzSTAR national facility at Swinburne, funded by Swinburne and the National Collaborative Research Infrastructure Strategy (NCRIS).

6:21PM P30.00006 Effect of fluid properties and impact speed for ophthalmic drug delivery via droplet impact. IDERA LAVAL, Texas Tech University, PEDRO MALLET, Universidade Federal Fluminense, JEREMY MARSTON, Texas Tech University — Front-of-the-eye (FOTE) delivery is the most popular route for administering drugs to the eye (i.e. ophthalmic delivery), with dropper bottles constituting a vast majority of the current market. These are notoriously inefficient and dictate the need for innovative devices to deliver discrete volumes to the eyeball. In addition, to promote adhesion many novel formulations incorporate polymers, leading to complex fluid properties. However, the fluid dynamics of this process and the specific role of fluid rheology combined with the pre-existing tear film are not well understood. Here, we will present preliminary results on delivery of polymeric solution using droplets, and discuss the effect of rheology, impact speed, substrate curvature, and evaluate universal scaling laws to capture the dynamics.

6:34PM P30.00007 Effect of applied load and jet dispersion on efficiency of needle-free injections. PANKAJ ROHILLA, JEREMY MARSTON, Texas Tech University — Intradermal delivery of vaccines with the jet injections is one of the leading alternatives to the needle injections for needle free drug delivery. However, effect of various parameters related to nozzle geometry, fluid properties and skin properties are still not well understood. In addition to design parameter such as the orifice diameter, jet speed, ampoule volume, and standoff distances, we must also consider applied load of the device on the skin, skin tension, and jet collimation. These three parameters are studied herein using an ex-vivo model (guinea pig and human skin). We investigate the effect of the skin support, viscosity of the liquid injected, ampoule volume, standoff distance and loading mechanism on the dispersion of the vaccine and the drug delivery efficiency into the skin.

1Inovio Pharmaceuticals, NSF-CBET

6:47PM P30.00008 Patient-specific analysis of esophageal transport using barium swallow fluoroscopy. SOURAV HALDER, Theoretical and Applied Mechanics, Northwestern University, SHASHANK ACHARYA, Mechanical Engineering, Northwestern University, WENJUN KOU, JOHN ERIK PANDOLFINO, PETER J. KAHRILAS, Feinberg School of Medicine, Northwestern University, NEELESH ASHOK PATANKAR, Mechanical Engineering, Northwestern University, Theoretical and Applied Mechanics, Northwestern University — Barium swallow is an X-ray test for diagnosis of esophageal disorders such as achalasia, diverticula, dysphasia, gastroesophageal reflux disease (GERD) etc. This test gives a qualitative idea of how efficiently a bolus is transported through the esophagus. We have developed a technique by which we can analyze a barium swallow fluoroscopy and calculate the velocity and pressure distribution inside the esophagus. In this technique, we use a Convolutional Neural Network to perform segmentation of image sequences generated from a fluoroscopy, and use these segmented images as input to a reduced-order model to calculate the velocity and pressure in the fluid inside the esophagus. We have also used this method along with High Resolution Manometry (HRM) to identify and estimate biomarkers such as wall-relaxation that occurs ahead of the peristalsis wave in the esophagus wall. This reduced-order model can run very fast, and hence can be used in clinical applications to add more quantitative information to the barium swallow test, thus resulting in better diagnosis of esophageal disorders.

1This work was supported by NIH grant 5P01DK117824-02.
7:00PM P30.00009 Flow in a cavity: fluid mechanics of kidney stone removal, JESSICA G. WILLIAMS, SARAH L. WATERS, DEREK E. MOULTON, BEN W. TURNERY, ALFONSO A. CASTRÉJÓN-PITA, University of Oxford — Flexible uretero-renoscopy provides a minimally invasive treatment for kidney stone removal. The ureteroscope is passed to the kidney through a hollow cylinder (access sheath) and has a central lumen (working channel) for surgical tools, such as laser fibres which pulverise stones. Resulting stone dust can impede the view from a miniscule camera at the scope tip; this necessitates irrigation – debris clearance by saline solution, which flows into the kidney through the working channel, and returns via the access sheath. Fast debris clearance allows for efficient ureteroscopy. We represent the renal pelvis of the kidney as a 2D rectangular cavity and investigate the effects of flow rate and cavity size on flow structure and subsequent clearance time. We model fluid flow with the steady Navier-Stokes equations, imposing a Poiseuille profile at the inlet boundary for the jet of saline, and zero-stress on the outlets, allowing for a parallel return flow. Resulting flow patterns in the cavity contain competing vortical structures. We demonstrate the existence of multiple solutions dependent on the Reynolds number of the flow and the aspect ratio of the cavity. We complement numerical predictions with PIV experiments. We model the clearance of an initial debris cloud via an advection-diffusion equation. We determine how the initial position of the debris cloud within the flow, and the Peclet number, affect clearance time, which is prolonged by entrapment of debris within closed streamlines. We discuss flow manipulation strategies to extract debris from vortices and decrease washout time.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P31 Biological Fluid Dynamics: Vesicle and Micelles I

5:16PM P31.0001 Patchy Vesicles Tremble Before a Flow, PRERNA GERA, University of Wisconsin-Madison, DAVID SALAC, University at Buffalo, The State University of New York, SAVERIO SPAGNOLIE, University of Wisconsin-Madison — Biological membranes, and recently engineered synthetic vesicles, may be host to numerous components which provide spatially varying material properties such as spontaneous curvature and bending rigidity. We will discuss the dynamics of a two-dimensional vesicle with such spatially varying material properties in a shear flow. Using small amplitude asymptotics and full numerical simulations, we pay special attention to the role of variable bending stiffness. Reduced-order models are derived and used to accurately predict phenomena ranging from low wavenumber breathing modes to highly oscillatory trembling modes.

5:29PM P31.0002 Non-contact Mechanical Characterization of Extracellular Vesicles with Raman Spectra Interpretations, JOANNA DAHL, University of Massachusetts Boston — Cells exchange information by secreting micro and nanosized extracellular vesicles (EVs) ranging in size from 30nm to 5um. While it was once thought these cell-derived membranous vesicles were simply cell debris, recent efforts have determined that EVs have profound biological significance and therefore potential for clinical therapies and disease diagnostics. There is still much to understand about fundamental EV biological, physical, and chemical properties before clinical applications can be developed. The mechanical behavior of EVs—the physical implications of the lipid, protein, and nucleic acid constituents and their arrangements, all of which are linked to EV biological signature, cell of origin, and mode of biogenesis—has hardly been explored. To date, EV mechanical properties have been measured with atomic force microscopy with its problematic adhesion and hard substrate effects for small, soft EVs. We present mechanical property measurements of single microscale EVs derived from human blood plasma using a non-contact microfluidic technique. Raman spectra are used to interpret the mechanical property measurements through analysis of protein and lipid composition and structures.

5:42PM P31.0003 Non-equilibrium Dynamics of Initially Spherical Vesicles in Shear Flows, AFSOUN RAHNAMA FALAVARIJANI, DAVID SALAC, University at Buffalo — Many vesicles have a spherical resting shape and exposure to shear flows induces an exchange between the suboptical/thermal fluctuations and system deformation, with the total area being conserved. Here, the dynamics of such vesicles is numerically explored. Unlike other models, we do not begin with a deflated vesicle. By taking into account the membrane fluctuations, our model allows for an increase in the apparent area of the vesicle which introduces an isotropic tension force on the membrane which grows exponentially with the change in the area in low tension regime. Our results, such as the viscosity-dependence of the tank-treading, breathing/trembling, and tumbling regimes, are in good quantitative agreement with experimental observations.

5:55PM P31.0004 Light-induced Non-pulsatile Bursting of Lipid Vesicles, VINIT KUMAR, University of Illinois at Urbana-Champaign, SANGWOO SHIN, University of Hawaii at Manoa, JIE FEŃG, University of Illinois at Urbana-Champaign — Lipid vesicles are topologically closed compartments bounded by semi-permeable lipid shells. Upon exposure to a hypertonic bath, vesicles respond with remarkable swell-burst events, showing oscillations in vesicle size and pore formation. Such lysis has been extensively harnessed to study dynamics of biological membranes, as well as releasing encapsulated actives for targeted drug delivery. Recently, some studies show osmotic shock by light-induced reactions achieves fast release of vesicle contents via an explosion process, e.g., irreversible bursting, yet the rationale for this is missing. Here we present a fundamental and quantitative understanding of bursting dynamics through a comprehensive theoretical model. Our model accounts for the kinetics of light-triggered reactions inside vesicles while considering the stochastic nature of pore nucleation by incorporating activation energy based on the vesicle expansion rate. The model quantitatively captures features of irreversible bursting dynamics, with good agreement between experimental observations and model predictions. Our work furthers a fundamental framework for nonequilibrium vesicle dynamics under osmotic stress induced by chemical reactions, offering design guidelines for vesicle-encapsulated substance release.
6:08PM P31.00005 Hydrodynamic stability of moderately-deflated, giant unilamellar vesicles in general linear flows. VIVEK NARSIMHAN, CHARLIE LIN, Purdue University — In this talk, we perform boundary element simulations to describe the shape and stability of osmotically deflated vesicles in a wide range of flows. The first half of this talk recaps vesicle dynamics in purely extensional flows, which are commonly found in contractions/expansions and/or suction flows. Above a critical flowrate, we find that moderately deflated vesicles undergo an asymmetric shape instability that looks fundamentally different than droplet breakup. The physical origins of such vesicle shapes are discussed in detail and compared with microfluidic experiments. In the second half of the talk, we discuss vesicle stability in a general linear flow that contains both vorticity and extension. We find that the critical capillary number for vesicle instability diverges as one moves from pure extension to pure shear, which suggests that vesicles are incredibly hard to break in pure shear flow. We also find that the vesicle’s interior viscosity plays little role in its stability, which is quite different than what is observed for droplets. We provide physical explanations for these observations by examining membrane tension profiles and using geometric scaling arguments. We will conclude by showing preliminary data of vesicle shapes in oscillatory flows.

6:21PM P31.00006 Break-up of synthetic capsule in shear flow. SEYEONG JEONG, DEEGYOUM KIM, KAIST — A capsule, a thin elastic membrane enclosing inner material such as colloid, has been used in diverse fields including drug delivery, cell encapsulation and cosmetics. The structural robustness of the capsule comes to an important issue for drug delivery because hydrodynamic shear stress acting on the membrane surface of the capsule in microcirculation environment may cause severe deformation and burst of the capsule. Previously, many studies have focused on the rheological behavior of the capsule immersed in simple flows such as shear flow, extensional flow or Poiseuille flow in small and moderate ranges of shear rate. However, little is known for the deformation of a capsule in high shear rate and furthermore its break-up phenomenon. For the break-up of a capsule in simple shear flow, experiment is performed with a flow rheoscope, and a capsule based on Human Serum Albumin is prepared. The capsule is modelled as a 2D thin shell by adopting the concept of hyperelasticity. The deformation and fracture of a capsule and the resultant stress distribution on the capsule surface are investigated by changing the mechanical properties of the capsule and the shear rate of the flow.

6:34PM P31.00007 ABSTRACT WITHDRAWN –

6:47PM P31.00008 Dynamics of highly deformed non-spherical vesicles in steady and time-dependent flow. DINESH KUMAR, CHANNING M. RICHTER, CHARLES M. SCHROEDER, Department of Chemical and Biomolecular Engineering, University of Illinois at Urbana-Champaign — In this work, we study the non-equilibrium dynamics of vesicles in precisely-defined steady and time-dependent extensional flow. Using Stokes trap, we directly observe non-equilibrium vesicle shapes as a function of reduced volume \( \nu \), viscosity contrast \( \lambda \), and Capillary number \( \text{Ca} \) using fluorescence microscopy. Vesicles are found to deform through a wide-range of interesting shapes in flow, including asymmetric and symmetric dumbbells, in addition to pearling, wrinkling, and buckling instabilities depending on membrane properties. Using this approach, we determine the flow phase diagram for vesicles in \( \nu-\text{Ca} \) space. Our results show that the steady-state deformation of vesicles exhibits power-law behavior as a function of reduced Capillary number. We identify two distinct relaxation processes for vesicles stretched to high deformation, revealing two characteristic time scales: a short time scale corresponding to bending relaxation and a long-time scale dictated by the relaxation of membrane tension. We further discuss the dynamics of single vesicles in sinusoidal oscillatory extensional flow as a function of \( \text{Ca} \) and Deborah number.

1This work was supported by the National Science Foundation by Award # NSF CBET 1704668.

7:00PM P31.00009 Collision of two deformable torque swimmers. HITOMU MATSUI, TOSHIHIRO OMORI, TAKUII ISHIKAWA, Dept. Finemechanics, Tohoku University — Understanding the property of a micro-organism suspension is important in bio-engineering. When the suspension is non-dilute, micro-organism interacts with each other and these interactions are governed by the hydrodynamical and biological features. Former studies focusing on cell-cell interaction have unveiled the hydrodynamical effect, however, contribution of cell’s deforming during the interaction is still unknown. Moreover, biological reaction, for example avoiding and escape reactions of a random effect, can react the suspension to affect the suspension to be initiated by mechanical stimuli imposing over cell membrane. Thus, analyzing membrane tension should help to understand the mechanism of ciliate biological responses. In this study, to investigate the contribution of deformation and membrane condition while two swimmers interact, we numerically simulate cell-cell interaction by applying a deformable ciliate model. We modeled a ciliate body as a deformable capsule and thrust forces generated by the ciliary beat as torque distribution. Owing to the tiny cell size of ciliate, fluid is regarded as Stokes flow. We computed a variety of collisions with different geometry and cell’s deformability, and then analyzed the trajectory and membrane tension. We found deformability affects the trajectory and clarified membrane tensions differ among the geometries. These results allow us to discuss the sensing ability of ciliate in the suspension.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P32 Biological Fluid Dynamics: General III

5:16PM P32.00001 Hydrodynamics of interphase chromatin. ACHAL MAHAJAN, University of California San Diego, WEN YAN. Center for Computational Biology, Flatiron Institute, New York, ALEXANDRA ZIDOVSKA, Center for Soft Matter Research, Department of Physics, New York University, MICHAEL J. SHELLEY, Center for Computational Biology, Flatiron Institute, New York and Courant Institute of Mathematical Sciences, New York University. DAVID SAINTILLAN, University of California San Diego. — Recent spectroscopy experiments on interphase chromatin have uncovered the existence of long-ranged coherent sub-diffusive motions on the scale of microns and persisting for seconds. These motions were found to be ATP-dependent suggesting the involvement of molecular motors. Motivated by these observations, we use Brownian dynamics simulations to elucidate the effects of microscale activity on the behavior and spatiotemporal dynamics of long flexible polymer chains in viscous solvents. We develop a coarse-grained model where active events are modeled as stochastic force dipoles, which drive long-ranged fluid flows inside an ellipsoidal nucleus. Numerical simulations based on a boundary integral formulation along with a kernel-independent fast multipole method demonstrate the key role played by hydrodynamic interactions and topological constraints in driving large-scale motions and chromatin reconfigurations.

1National Science Foundation Division of Engineering (CMMI-1762566) and Extreme Science and Engineering Discovery Environment (XSEDE), National Science Foundation grant number ACI-1548562.
5:29PM P32.00002 Application of a High-Speed Plenoptic Camera for 3D Measurements in Small-Scale Biological Flows  
ZU PUAYEN TAN, RICHARD ALARCON, JOHANNES ALLEN, BRIAN S. THUROW, ANTHONY MOSS, Auburn University, ADVANCED FLOW DIAGNOSTICS LAB TEAM, MOSS BIOLOGICAL SCIENCES LAB TEAM — The application of conventional multi-camera tomographic-PIV and related 3D techniques to study biological flows remains limited due to their expense, complexity and bulk. This is particularly true for small experiments (e.g., heart-valve and micro-swimmer) or portable setups (e.g., carried by divers in open-water measurements) where equipment footprint is a critical constraint. In this presentation, we propose single-camera plenoptic-PIV as an attractive alternative to the standard multi-camera techniques. Specifically, a modular kHz-rate plenoptic camera newly developed at Auburn University will be introduced. The system, composed of a single main-lens, a microlens adaptor and an off-the-shelf high-speed camera, was used to characterize unsteady 3D flows around a 2cm ctenophore Mnemiopsis in a 71x40x34mm volume. Various 3D flow features such as the creature’s downwash and intermittent vortex-ejections were successfully captured. These will be presented to showcase the plenoptic system’s capabilities for measuring small-scale biological flows.

5:42PM P32.00003 Visualizing flow inside a bone porous medium using an MRI machine 1, 2, 3, 4  
SUUYE HAN, TODD CURRIER, University of Massachusetts Amherst, MAHDIAR EDRAKI, Northeastern University, BOUYAN LIU, University of Massachusetts Amherst, MAUREEN LYNCH, University of Colorado Boulder, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — We have used Phase-Contrast Magnetic Resonance Imaging (PC-MRI) flow measurement to quantify flow inside a 3D-printed artificial scaffold model to understand the flow behavior inside the 3D model of bone metastasis due to an applied perfusion. In order to perform the test using an MRI machine, a nonmagnetic water tunnel was designed and built. A 3D surface model created from a micro-CT scan of an artificial scaffold model was used to make the 3D-printed scaffold model. The 3D-printed scaffold was placed in the test section of the water tunnel inside the MRI machine. The flow velocity was varied over a range and images were captured using the MRI machine. The phase and magnitude data from the MRI experiment were then processed using an in-house code to quantify the flow inside the scaffold.

5:55PM P32.00004 A mathematical framework for developing freezing protocols in cryopreservation  
MOHIT DALWADI, SARAH WATERS, HELEN BYRNE, IAN HEWITT, University of Oxford — Cryopreservation is the process of preserving biological constructs by cooling to temperatures low enough to halt biochemical processes, such as metabolism. This allows biomaterials to be kept in 'suspended animation', with important applications in tissue engineering, fertility, and food security. However, many freezing protocols have low recovery rates. In general, cooling too quickly results in the formation of lethal intracellular ice, while cooling too slowly amplifies the toxic effects of the cryoprotective agents (CPA) added to limit ice formation. In this talk, we present a mathematical model for cryopreservation to understand and quantify these observations. We consider a system consisting of three different regions: ice, extracellular liquid, and intracellular liquid. The two interfacial boundaries separating the three phases can move and must be determined as part of the solution. The presence of CPA lowers the freezing point of the system, and the cell membrane moves due to the osmotic pressure difference across the membrane. We introduce two metrics to characterize the cell damage caused by freezing, accounting for supercooling and CPA toxicity. Given cell properties, we show how these damage metrics can be used to predict an optimal cooling rate.

6:08PM P32.00005 Characterization of mucus macro-rheology from the silver carp, Hypophthalmichthys molitrix 1, 2  
KARTIK V BULUSU, SAMANTHA RACAN, L PATRICIA HERNANDEZ, MICHAEL W PLEŚNIAK, The George Washington University — The silver carp, Hypophthalmichthys molitrix is a planktovorous filter feeder fish introduced to control algal blooms in natural waterways of the US. Since the early 1970s this invasive species has infested the Mississippi River basin. Its extraordinary feeding-efficiency is attributed to two unique organs viz., (i) the gill rakers (GR) and (ii) the palatal folds that enable capturing of food particles through the porous GR membranes. The GR mucus has the potential to enhance the filter feeding process by functioning as an adhesive and a transport vehicle for food particles. It is a gel-like, complex biological fluid that responds to external force and comprises a macromolecular network of glycoproteins (or mucins). Viscoelasticity and steady-state viscosity of the GR mucus of silver carps obtained from Hart Creek, Missouri River were investigated using a rheometer (DHR-2, TA Instruments) with cone geometry (1-deg., 40 mm dia.) and a Peltier plate. A digital camera attachment (546800.902, TA Instruments) was used to monitor microstructure changes. These experiments are aimed at understanding the role of mucus-laden fluid flow through porous GR channels and ultimately, the tremendous success of the silver carp in outcompeting native fish species.

6:21PM P32.00006 Self-organization of microtubules in cell-sized droplets 1  
YA GAI, Mechanical and Aerospace Engineering, Princeton University, SAGAR SEtru, Lewis-Sigler Institute for Integrative Genomics, Princeton University, BERNARDO GOUVEIA, HOWARD STONE, Mechanical and Aerospace Engineering, Princeton University, SABINe PETRY, Molecular Biology, Princeton University — We combine droplet microfluidics and cell-free biological systems to examine the effect of confinement and nucleation on the assembly of microtubule (MT) networks. Central to the spindle assembly is the spatial organization of MTs, a long tubular structure formed through the polymerization of tubulin dimmers. Such organization is regulated by RanGTP, a GTPase associated with chromosomal activities and acting as part of a major nucleation pathway for MTs. RanGTP has been explored using Xenopus egg extracts, a model cell-free system for probing spindle assembly. Most extract-based assays were performed in a test tube where cell-sized confinement was missing. Therefore, we asked whether confinement can affect the MT networks. We used droplet microfluidics for encapsulating extract-based assays by generating monodisperse, extract-in-oil droplets. By varying droplet diameters and encapsulated Ran concentrations, we demonstrate that these two physical factors regulate the assembly of MT networks. Together, the two factors yield MT networks with various steady-state architectures. Our results highlight the prominent role of MT nucleation in the self-organization of MTs in cell confinement and might have direct implications in nucleation-controlled soft matter processing.

6:34PM P32.00007 Stokes’ law in complex liquids and inside cell cytoplasm 1  
KAROL MAKUCH, California Institute of Technology, ROBERT HOLYST, TOMASZ KALWARCZYK, PIOTR GARSTECKI, Institute of Physical Chemistry, Polish Academy of Sciences, JOHN F. BRADY, California Institute of Technology — The ‘viscosity’ experienced by a small tracer particle in complex liquids depends both on its size and on the structure of the liquid, which itself may contain different length scales. Thus, in a microenvironment the complex liquid may best be described by wave-vector-dependent viscosity $\eta(k)$. Here we derive Stokes’ law in complex liquids and formulate a method to determine the wave-vector-dependent viscosity from microrheological experimental data. We initiate our approach by determining the wave-vector-dependent viscosities $\eta(k)$ of HeLa and Escherichia Coli cell cytoplasm from the experimental data on diffusion of macromolecules in these systems. Determination of this quantity opens an avenue for computer simulations of motion and biochemical reactions inside living cells.

---

1Supported by GW Center for Biomimetics and Bioinspired Engineering

2Princeton Catalysis Initiative
6:47PM P32.00008 Freshwater Copepod Behavior in Turbulent Eddies. M. RUSZCZYK, D.R. WEBSTER, J. YEN, Georgia Tech — Previous studies have shown that marine copepod behavior is modulated by turbulence. We seek to expand on this observation by investigating a freshwater species, *Hesperodiaptomus shoshone*, and how it responds to small-scale turbulent vortices. The calanoid copepod *H. shoshone* is a dominant predator in high-altitude alpine lakes, ranging from 2-4 mm in length. The Burgers vortex model was used to simulate dissipative eddies with four levels of turbulent dissipation rates ranging from 0.002-0.25 cm$^2$/s$^3$, mimicking turbulent conditions found in natural habitats. Tomographic PIV was used to quantify the vortex circulation and axial strain rate of the vortices. *H. shoshone* males and females were separately exposed to the four Burgers vortex treatments in a horizontal axis orientation plus a stagnant fluid control treatment. In comparison to the morphologically-similar marine copepod *Calanus finmarchicus*, *H. shoshone* appears minimally responsive to the Burgers vortex. *H. shoshone* swimming speeds remained similar under different turbulence conditions, showed no circular trajectories, and made minimal attempts to escape the vortex. These results were consistent between males and females and suggest *H. shoshone* are not responsive to hydrodynamic cues of the vortex structure. The contrasting behavior will be discussed in the context of the ecology and environmental conditions of the different habitats.

7:00PM P32.00009 Computational Investigations on Flow-mediated Transport Processes at the Blood-thrombus Interface. DEBANJAN MUKHERJEE, University of Colorado Boulder — Pathological clotting of blood, referred to as thrombosis, is the primary cause of diseases like stroke and heart-attack which are associated with significant morbidity and mortality. Thrombus (blood clot) formation, and its behavior, is governed by multiple underlying physiological processes which are intimately related to flow and transport. Of specific importance is the role of transport processes near the blood-thrombus interface, and permeation of biochemical species (drug or coagulation factors) into the thrombus. Here we present the latest developments in our investigations into creating a computational multi-physics modeling framework for thrombus biomechanics and bio-transport. Our framework is based on a combination of Galerkin stabilized finite element method, and Lagrangian particle based approach. We have employed our framework to conduct a range of computational experiments to illustrate flow mediated near thrombus transport using thrombus models reconstructed from microscopy image data. We will present results from our computational investigations into: (a) the role of blood-thrombus interface properties on transport; (b) biochemical species permeation across the thrombus interface; and (c) subsequent influence on intra-thrombus transport.

This work was partly supported by American Heart Association (AHA:16POST27500023)

Monday, November 25, 2019 5:16PM - 7:00PM — Session P33 Flow Instability: Boundary Layers 615 - Philipp Hack, Stanford University

5:16PM P33.00001 High-speed linear analysis of a blunt cone. TIM FLINT, PARVIZ MOIN, M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University — Transition to turbulence critically affects heat transfer towards the walls in the flow over high-speed vehicles. We seek to identify the mechanisms which govern the receptivity and amplification of disturbances in the boundary layer on a blunt cone at Mach 6 by means of global linear analysis. The approach directly captures the influence of the bow shock wave as well as the geometry of the blunt nose of the cone. Two families of modes are computed, and their spatial structure is characterized. One group of modes extend beyond the boundary layer and interacts with the vortical post-shock flow near the cone tip. The other describes acoustic emissions and decays in the free-stream. The receptivity of the modes is directly captured in their corresponding adjoint eigensolutions. receptivity is highest upstream of the bow shock near the cone axis. The computations are performed using a curvilinear framework with shock capturing and continuous, discretely consistent formulations of the linearized and adjoint linearized governing equations.

This work is supported by the Office of Naval Research under grant N00014-17-1-2341.

5:29PM P33.00002 Study for Wall-shear Stress of Pulsatile Flows in 3-D Ducts. XIAOYU ZHANG, SYEEDALIREZ ABOOTORABI, Mechanical and Energy Engineering, Indiana University-Purdue University, Indianapolis, HIROKI YOKOTA, Biomedical Engineering, Indiana University-Purdue University, Indianapolis, INDIANPOLIS, HUIDAN YU, Mechanical and Energy Engineering, Indiana University-Purdue University, Indianapolis — Understanding wall shear-stress (WSS) in human tissues and organs such as blood vessels and synovial bursa is critically important in the prevention, pathogenesis, and treatment of varying diseases. Numerical simulation provides a unique tool for a fast and non-invasive quantification of WSS in realistic flows. We use a GPU-accelerated volumetric lattice Boltzmann method (VLBM) to assess WSS of pulsatile flows in 3D ducts with circular and rectangular cross sections. The computational results are validated through the comparisons with analytical solutions of Womersley flows in the two ducts. From the analytical solutions of Womersley flow, driven by pressure gradient $\partial p/\partial r=P_0+P_0 e^{i\omega}$, we observed that the WSS is linear to the magnitude of the unsteadiness ($P_0/P_0$). Preliminary analysis indicates that the WSS is promoted/suppressed with small/large Womersley number. The effects of pulsation in more realistic pulsatile pipe flows with and without turbulence will be further examined and presented.

The research is supported by NSF grant (CBET 1803845) and IUPUI University Fellowship. The Extreme Science and Engineering Discovery Environment (XSEDE), supported by National Science Foundation grant number ACI-1053575, was used.

5:42PM P33.00003 Hypersonic boundary layer transition over curved-walls: A mechanism based on Gorlter vortices. ANUBHAV DWIVEDI, University of Minnesota, GS SIDHARTH, Los Alamos National Lab, GRAHAM V CANDLER, University of Minnesota, MIHAILO R JOVANOVIC, University of Southern California — We investigate amplification of small disturbances in compressible boundary layers on a flat plate with a concave flare. To understand mechanisms that trigger the transition in the boundary layer flow over a curved wall, we utilize input-output analysis to quantify the receptivity of flow fluctuations to exogenous disturbances. Our analysis identifies Gorlter vortices as the most amplified flow structures and provides insights into mechanisms that select the dominant spanwise wavelength. The effect of wall heat transfer on the growth of boundary layer perturbations is also explored. Furthermore, we complement the input-output analysis with direct numerical simulations to investigate the non-linear stages of the disturbance evolution. Since Gorlter vortices are often responsible for boundary layer transition on curved walls, methods to attenuate early stages of their amplification are also analyzed and their effectiveness is discussed.

Office of Naval Research (grant numbers: N00014-16-1-2500 and N00014-19-1-2037)
the effect of the competition on the onset of instability. The results are verified by performing the experiments and non-linear simulations.

Relative to the time dependent base state using momentary stability. Growth rate is adequately defined to take into account the radial source

discussion. The LSA provides an alternate approach to study time dependent linear system arising in

displacement of a more viscous fluid by a less viscous one in a porous medium or Hele-Shaw flow. We perform a state-of-the-art linear stability

MANORANJAN MISHRA, VANDITA SHARMA, Indian Institute of Technology Ropar, INDIA, SATYAJIT PRAMANIK, University of Oxford,

supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

Brown University, Presidential Fellowship.

Interfacial instability in an azimuthally oscillatory two-layer fluid of oil and water1. LINFENG PIAO, HYUNGMIN PARK, Seoul National University — There are interfacial instabilities occurring in multi-layer flows, which are of practical importance in many industrial processes, such as coating, solvent extraction and oil recovery. Especially, these instabilities could be significantly influenced by the external perturbation, e.g., oscillating forcing. In this study, we present experimental results concerning the stability of oscillatory two-layer fluid in a vertical cylinder vessel, using a high-speed imaging. Two immiscible fluids (oil and water) with a relatively low viscosity contrast (~100), are superposed in the vessel and the oscillating frequency and angular amplitude are varied by 0.1-7.0 Hz and 45-180 degrees, respectively. We measure the evolution of the oil-water interface during the oscillation and identify the wavenumber of the disturbances is much smaller than both wall-normal and spanwise wavenumbers. The base flow is excited either by freestream disturbances imposed at the upstream boundary or by disturbances from the wall in the form of wall transpiration. An extensive parametric study is performed in different flow conditions to assess the development of these streaky structures.

Nonlinear spatial marching of high-amplitude perturbations1. SHAUN R. HARRIS, M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University — The physics of bypass transition are closely related to the formation of highly energetic streaks within the boundary layer. Owing to the high amplitudes of the streaks, nonlinear interactions and the generation of harmonics play an important role in their development, as well as in triggering their secondary instability. The high amplitudes attained by the streaks also pose unique challenges for their computational modeling. While the nonlinear parabolized stability equations have been shown to accurately describe the exponential amplification of disturbances in classical transition scenarios, they inadequately capture the formation of streaks and the associated generation of perturbation harmonics. In this talk, we present a novel framework for the nonlinear spatial marching of high-amplitude perturbations. In comparisons with direct numerical simulations, we demonstrate the accuracy of the approach in predicting the growth of high-amplitude streaks. The computational cost of the method is comparable to that of the established nonlinear parabolized stability equations.

Session P34 Flow Instability: Interfacial and Thin Film Fingering 616 - Dominique Legendre, IMFT

5:16PM P34.00001 Linear Stability Analysis of Radial Miscible Viscous Fingering . MANORANJAN MISHRA, VANDITA SHARMA, Indian Institute of Technology Ropar, INDIA, SATYAJIT PRAMANIK, University of Oxford, CHING-YAO CHEN, National Chiao Tung University, Taiwan — Viscous fingering (VF) is a hydrodynamic instability ubiquitous during the displacement of a more viscous fluid by a less viscous one in a porous medium or Hele-Shaw flow. We perform a state-of-the-art linear stability analysis (LSA) of miscible VF with a radial flow. The LSA provides an alternate approach to study time dependent linear system arising in miscible VF. The time dependent base state is calculated numerically using the method of lines. The evolution of the disturbances is studied relative to the time dependent base state using momentary stability. Growth rate is adequately defined to take into account the radial source flow. The LSA captures the competition between advective and diffusive forces during the initial stages of radial VF and gives an insight into the effect of the competition on the onset of instability. The results are verified by performing the experiments and non-linear simulations.
5:29PM P34.00002 Immiscible Fingering Instability via Alternating Injection: Experiments and Simulations
CHI-CHIAN CHOU, WEI-CHENG HUANG, CHING-YAO CHEN, Department of Mechanical Engineering, National Chiao Tung University, Taiwan, R.O.C. — Viscous fingering instability on an immiscible interface via alternating injection is investigated by both experiments and numerical simulations. Multiple fluid annuluses associated with unstable and stable interfaces are resulted by injecting the less and more viscous fluid alternatively. We focus on the influences of two control parameters, e.g., the alternating injection interval and viscosity contrast, to the development of fingering instability. It is interesting to observe less prominent instability, determined by the injection interval and the viscosity contrast, as the interface grows. The results are inconsistent with the conventional continuous injection. This inconsistent behavior is mainly due to rupture of the fluid annuluses of the less viscous fluid. Because of surface tension, the ruptured less viscous fluids tend to form separated drops, so that the total interfacial length decreases. On the other hand, the fluid annuluses appear more stable in the cases of lower viscosity contrast, so that the annuluses are stretched to prolong the overall interfacial length.

1Support by the R.O.C. (Taiwan) MOST through the grant 107-2221-E-009-070-MY3 is acknowledged.

5:42PM P34.00003 Pattern Formation from Instabilities in Chromonic Liquid Crystals
IRMGARD BISCHOFFBERGER, QING ZHANG, MIT, SHUANG ZHOU, UMass Amherst — The displacement of a more viscous fluid by a less viscous one in a quasi-two-dimensional geometry leads to the formation of complex fingering patterns. In isotropic systems, disordered dense-branching morphologies arise from repeated tip-splitting of the evolving finger. In anisotropic systems, by contrast, the growth morphology changes to a highly ordered dendritic growth characterized by stable needle-like protrusions decorated with regular side-branches. We investigate such morphology transitions between dendritic growth and dense-branching growth in an intrinsically anisotropic liquid; a lyotropic chromonic liquid crystal in the nematic phase. We show that the transition is remarkably sensitive to the interface velocity and the viscosity ratio between the less-viscous inner fluid and the more-viscous outer liquid crystal. We discuss the importance of a stable shear alignment of the liquid crystal in governing the morphology transition to dendritic growth.

5:55PM P34.00004 Rivulet formation in falling liquid films
GIANLUCA LAVALLE, JULIEN SEBILLEAU, DOMINIQUE LEGENDRE, Institut de Mécanique des Fluides de Toulouse (IMFT), Toulouse, France — When a thin liquid film falls down an inclined or vertical plane, a capillary ridge develops behind the advancing contact line. For sufficiently thick ridges, rivulets appear as a result of the contact-line instability. The formation of such complex liquid structures might degrade the performances of several industrial applications, such as coating, aerodynamic efficiency and chemical processes. We investigate the dynamics of liquid films of partially wetting fluids falling down a vertical plane. We are particularly interested in the fingering instability and the distribution of rivulets. Based on direct numerical simulations, we study the influence of initial contact-line perturbations and pre-existing surface contaminations upon the fingering topology. Interestingly, varying the wave-number of the initial sinusoidal contact-line perturbation leads to different rivulet topologies and root velocities. Meanwhile, the presence of a pre-existing sufficiently large drop forces the formation of a rivulet where the drop and the contact line interact, thus destroying the symmetry of the flow. Finally, we discuss the wetted area for several drop arrangements.

5:55PM P34.00005 Stability and fingering of radial flows
JOHN LISTER, FREDERIK DAUCK, University of Cambridge — A pool of honey spreading on a horizontal surface becomes axisymmetric, but radial flow in a Hele-Shaw cell is unstable to fingering if the injected fluid is sufficiently less viscous than the ambient. Why the difference? Radial geometries offer a distinct advantage for the analysis of fingering instabilities of spreading flows in that the azimuthal wavenumber remains constant and thus self-similarity methods can be employed. Several problems of this sort are described. For example, it is shown analytically that viscous gravity currents, with power-law injection and power-law flux relationship $q = -h^n \nabla h$, are stable. And it is shown by analysis of a self-similar kinematic wave that Hele-Shaw flows of miscible fluids at infinite Peclet number are unstable if and only if the viscosity ratio exceeds 3/2.

6:21PM P34.00006 Fingering Instabilities in Oxidizing Eutectic Gallium-Indium
KEITH HILLAIRE, North Carolina State University, WILLIAM LLANOS, Chicago State University, MICHAEL DICKEY, KAREN DANIELS, North Carolina State University — Eutectic gallium-indium (eGaIn), a room-temperature liquid metal alloy, has the largest tension of any liquid at room temperature, and yet can nonetheless undergo fingering instabilities. This effect arises because, under an applied voltage, an oxide builds up on the surface of the metal. The oxide acts like a surfactant, lowering the surface tension and allowing spreading under gravity. In the experiments described here, we examine the hypothesis that fingering instabilities, including tip-splitting, arise due to Marangoni instabilities. Our experiments are performed with eGaIn droplets placed in an electrolyte bath of sodium hydroxide; by placing the eGaIn on copper electrodes, which eGaIn readily wets, we are able to impose a fingering wavelength on the spreading. Two transitions are observed as a function of current: (1) a minimum current at which eGaIn spreads out from the copper electrode; (2) the current at which the fingers become unstable to shorter wavelengths and spread inhomogeneously. We present a phase diagram as a function of current and initial wavelength, and identify a minimum wavelength below which single tip-splitting does not occur.

6:34PM P34.00007 Delayed onset in viscous fingering
THOMAS VIDEBAEK, SIDNEY NAGEL, Department of Physics, University of Chicago — The viscous fingering instability occurs at the evolving interface between two viscous fluids confined to a thin gap. We investigate the onset of the instability in both radial and rectangular geometries for both miscible and immiscible pairs of fluids. In all four cases we observe a region of stable growth, $L_{\text{stable}}$, before fingers start to develop. While the initial stability in the radial cell has been ascribed to the velocity profile associated with point-source injection, this explanation is much too small to explain our observations. The region of stable growth before onset that we observe in the linear cell is unexpected. For miscible fluids, $L_{\text{stable}}$ can be tied to the distance it takes to form steady-state, interfacial structures in the gap. For immiscible fluids, where no internal structure is apparent, there is no obvious explanation for $L_{\text{stable}}$, which we find also depends on the capillary number. These results are not accounted for by current analyses of fingering dynamics. We suggest that this is because the experiments are by nature always quasi-two dimensional with an important length set by the gap size.

6:47PM P34.00008 Stabilization of viscous fingering in a partially miscible system
RYUTA SUZUKI, SHOJI SEYA, Tokyo University of Agriculture and Technology, TAKAHIKO BAN, Osaka University, MANORANJAN MISHRA, Indian Institute of Technology, Ropar, YUCHIRO NAGATSU, Tokyo University of Agriculture and Technology — Viscous fingering (VF) or Saffman-Taylor instability occurs when a less viscous fluid displaces a more viscous one in porous media or in Hele-Shaw cells. The classical VF can be divided into two; miscible and immiscible systems depending on whether two fluids are miscible or immiscible. In addition, it has been recently reported that a partially miscible VF has experimentally shown to change to multiple droplets pattern. However, in the present study, we have experimentally shown a partially miscible VF can have the potential to stabilize the interface more effectively, namely, leading to circular-like pattern in a radial geometry. This is considered to be caused by the two factors; a convection induced by spinodal decomposition directed from the more viscous fluid to the less viscous one and a high rate of the spinodal decomposition.
fingering formed by the injection of Newtonian surfactant solution can be used to contribute to the establishment of well-controlled processes for surfactant flooding and the recovery of residual NAPL in aquifers.

However, in the present study, phenomena contrary to these rules were observed: wider fingers occurred in the surfactant solution system compared to those in the water system, in the nonlinear stage of VF evolution, despite the fact that the capillary number had the same value for both systems. In addition, even though the surfactant system had a higher capillary number than the water system, wider surfactant fingers were observed. A possible mechanism explaining this is discussed by comparing with previous studies regarding VF with surfactants. The present study indicates that the capillary number does not control the nonlinear VF width in the surfactant system. Our results and discussion can be used to contribute to the establishment of well-controlled processes for surfactant flooding and the recovery of residual NAPL in aquifers.

Monday, November 25, 2019 5:16PM - 7:13PM
Session P35 Microscale Flows: Active Particles 617 - John Buchner, TU Delft

5:16PM P35.00001 MicroPIV measurements of flows induced by achiral microswimmers
JAMEL ALI, FAMU-FSU College of Engineering, LIYUAN TAN, XIANGCHENG SHI, Southern University of Science and Technology, DALHYUNG KIM, Southern Illinois University Carbondale, MIN JUN KIM, Southern Methodist University, U KEI CHEANG, Southern University of Science and Technology — We report on the low Reynolds number hydrodynamics induced by achiral microswimmers and analysis of their flow fields using particle image velocimetry. The flows produced by two types of rigid swimmers are examined. The first swimmer consists of three self-assembled magnetic beads bonded together with avidin-biotin complexes. The second swimmer consists of thin geometries in the shape of the letter ‘L’ produced through photolithography and thin-film deposition. Both swimmers were wirelessly actuated in precessing magnetic fields using electromagnetic coils positioned in an approximate Helmholtz configuration. A high speed camera was used to capture the motion of fluorescent seeding particles as well as track the rotation of the microswimmers. Analysis of microPIV data revealed microvortices produced during swimming, while the magnitude of the local flows scaled linearly with increasing rotation rate. The attractive and repulsive flows observed may be useful for applications such as non-contact micro manipulation and assembly/disassembly of modular swimmers.

1This work was funded by the National Natural Science Foundation of China (NSFC; 1850410516), Department of Education of Guangdong Province (2017KTSCX167), and Shenzhen municipal government (Peacock Plan, 20181119590C).

5:29PM P35.00002 Templated self-assembly of colloidal microswimmers
ANTOINE AUBRET, JEREMIE PALACCI, University of California San Diego — Biological living systems are prototypical examples of Active Matter. Cells, for instance, exist far from equilibrium behavior such as autonomous regulation or organization. Here, we show how we can carve non-equilibrium pathways for the controlled self-assembly of colloidal microswimmers using light as a tool. We use photocatalytic colloidal microparticles as primary building blocks for self-assembly. We specifically designed the particles to self-propel, and sense light gradients. Following sequential light-patterns, the particles autonomously assemble into robust self-spinning structures, or microgears. The gears interact with contactless ‘teeth’, synchronizing their motion. We characterize the interaction potential, and show that the synchronization originates from the coupling between the chemical clouds generated by the catalytic activity of the gears and the hydrodynamic interactions between their constituents. Following, the gears constitute the fundamental components of synchronized micro-machineries that auto-regulate and whose dynamics is tuned by the spins of their internal components. Our study demonstrates the potential of non-equilibrium interactions to program self-assembly of dynamical colloidal architectures.

5:42PM P35.00003 Steady streaming in a simple reciprocal swimmer
NICHOLAS DERR, CHRISTOPHER RYCROFT, Harvard University, DAPHNE KLOTSZA, University of North Carolina at Chapel Hill — While biolocomotion at high and low Re has been well-studied, swimming at intermediate Re ∼ 1–1000 — where both viscous and inertial forces are important — is less understood. Most previous investigations at intermediate Re have centered on individual species, implicitly focusing on a single example of the many ways such organisms self-propel. As a result, few underlying generic mechanisms that unify the many disparate intermediate-Re swimming methods have been identified. One possible such mechanism is steady streaming — the generation, due to inertial effects, of lower-order steady flow by periodic large-scale motion. In this talk, we examine the role of steady streaming in the locomotion of a simple reciprocal swimmer at intermediate Re. After asymptotically expanding the Navier–Stokes equations, we solve for time-periodic solutions to the resulting set of unsteady Stokes equations. We present the swimming speed and efficiency over a range of intermediate Re and comment on similarities to motility mechanisms at low Re.

1ND acknowledges support from DoD through the NDSEG fellowship, the NSF-Simons MathBio Center at Harvard (award number #1764269), and the Harvard QBio Initiative

5:55PM P35.00004 Life in the fast layer
ERNEST B. VAN DER WEE, Department of Physics & Astronomy, Northwestern University, BRENNAN SPRINKLE, Courant Institute of Mathematical Sciences, New York University, ISAIAH KATZ, Department of Physics & Astronomy, Northwestern University, MENA YOUSSEF, STEFANO SACANNA, Department of Chemistry, New York University, ALEKSANDAR DONEV, Courant Institute of Mathematical Sciences, New York University, MICHELLE M. DRISCOLL, Department of Physics & Astronomy, Northwestern University — Microrollers are rotating particles that become active close to a wall due to an asymmetric flow of the fluid around the particles. They can be experimentally realized by driving magnetic colloidal particles hovering above a wall with a rotating magnetic field. Introducing a small fraction of fluorescently labeled microrollers, we can measure their velocities using microscopy and particle tracking. We compare our results to high resolution Brownian dynamics simulations which include lubrication effects. The velocity of a microroller is much slower than the velocity of the fluid pumped around it. Therefore, a particle put in the flow field around a microroller will have a velocity much higher than the microroller itself. As a consequence, the average velocity of a suspension of microrollers increases as a function of their density. In addition, at higher densities the particles form two layers: a slow one close to the wall and a much faster one above it. We find that the microrollers switching between the slow and the fast layer, and characterize the lifetime of the particles in the two layers.
This project has received funding from the European Research Council (ERC) under the European Unions Horizon 2020 research and innovation programme under Grant Agreement 714027 (SM).

6:08PM P35.00005 Field flow on the interface induced by swimming and driven colloids1, MEHDI MOLAEI, JIA DIYI DENG, TIANYI YAO, NICHOLAS CHISHOLM, JOHN CROCKER, KATHLEEN STEBE, University of Pennsylvania — An active colloidal trapped on or near a fluid interface generates a complex flow field that differs from the bulk flow. The flow field depends on the modes of motion, the mechanics of the interface, and hydrodynamic coupling with the bulk fluids. To characterize the flow field, we introduce a method based on the correlated motion of active colloids and passive tracer particles. The challenge is to extract weak biased motions of probes via interaction with the active colloids given the noisy environment and significant Brownian displacement. We examine a gallery of active motions and their flow fields; we simultaneously measure the rheology of the interface. We first investigate a 2D bacterial suspension on an oil-water interface as a model active colloidal system. We analyze the flow field induced by pusher and puller bacteria (models of force dipoles) and “piroetting” bacteria (stationary rotlet dipoles) and driven magnetic microbeads (Stokeslets). The measurements are performed for the interfaces with different viscoelasticity. Tracer particle displacement fields at various lag times are compared to calculated displacement fields for hydrodynamic modes permitted in interfacial layers as a function of rheology and compressibility.

1GoMRI

6:21PM P35.00006 Relating microswimmer synthesis to hydrodynamic actuation and rheotactic tunability1, ENKELEIDA LUSI, Department of Mathematics, New Jersey Institute of Technology, QUENTIN BROSSEAU, Courant Institute, New York University, FLORENCIO BALBOA USABIAGA, Flatiron Institute, YANG WU, New York University, LEIF RISTROPH, Courant Institute, New York University, JUN ZHANG, MICHAEL WARD, MICHAEL J. SHELLEY, New York University — We explore the behavior of micron-scale autophoretic Janus (Au/ Pt) rods, having various Au/Pt length ratios, swimming near a wall in an imposed background flow. We find that their ability to robustly orient and move upstream, i.e. to rheotax, depends strongly on the Au/Pt ratio, which is easily tunable in synthesis. Numerical simulations of swimming rods actuated by a surface slip show a similar rheotactic tunability when varying the location of the surface slip versus surface drag. Slip location determines whether swimmers are Pushers (rear-actuated), Pullers (front-actuated), or in between. Our simulations and modeling show that Pullers rheotax most robustly due to their larger tilt angle to the wall, which makes them responsive to flow gradients. Thus, rheotactic response infers the nature of difficult to measure flow-fields of an active particle, establishes its dependence on swimmer type, and shows how Janus rods can be tuned for flow responsiveness. We demonstrate the effectiveness of a simple geometric sieve for rheotactic ability.

1This was supported by NSF-MRSEC program DMR-1420073, and NSF Grants DMS-1463962 and DMS-1620331.

6:34PM P35.00007 Mobility mechanisms condition the instabilities in active microdroplets1, MATVEY MOROZOV, SEBASTIEN MICHELIN, Ecole Polytechnique — Chemically active droplets submerged in the bulk of surfactant solution self-propel with straight, helical, or random trajectories. Here we employ numerical simulations to establish the link between the behavior of an active drop and its interfacial properties. To this end, we consider a drop that converts the gradients of surfactant concentration into flow via two different mobility mechanisms: diffusiophoresis and the Marangoni effect. The resulting surfactant advection is the only nonlinear effect and, thus, the only source of dynamical complexity in the model. Our numerical simulations indicate that strong advection may destabilize the regime of straight and steady self-propulsion. For axisymmetric flow, this instability results in a regime of symmetric extensile flow around a stationary droplet. If advection is strengthened further, chaotic oscillations may develop. In 3D, a drop driven by diffusiophoresis alone does not exhibit extensile flow and the random behavior emerges right after the steady self-propulsion becomes unstable. Our results reveal that the thresholds of these instabilities depend heavily on the balance between diffusiophoresis and the Marangoni effect.

1Funded by ERC grant no. 714027 to S.M.

6:47PM P35.00008 Direct Numerical Simulations of Electric Field Driven Hierarchical Self-assembly in Mixtures of Particles1, S. B. PILLAPAKKAM, Associate Professor, SUCHANDRA DAS, Student, EDISON AMAH, Research Engineer, IAN FISCHER, PUSHPENDRA SINGH, Professor — We have numerically studied the process of self-assembly in particle mixtures when they are subjected to an externally applied electric field. The inter-particle electric forces cause mixtures of micron to nano sized particles to self-assemble into molecular-like hierarchical arrangements consisting of composite particles which are organized in a pattern. As in experiments for micron sized particles, the structure of a composite particle depends on factors such as the relative sizes and the number ratio of the particles, their polarizabilities, and the electric field intensity. The minimum electric field intensity required for manipulation is larger for nanoparticles for which the electric field induced lateral forces must also overcome Brownian forces. Also, for nanoparticles, the composition of composite particles was relatively more uniform because of the mixing induced by Brownian motion. Particles of mixtures containing only positively or negatively polarizable particles arrange in chains and columns which become aligned in the electric field direction, but when both type of particles are present they come together to form clusters.

1National Science Foundation

7:00PM P35.00009 Collisions and rebounds of active droplets1, KEVIN LIPPERA, MATVEY MOROZOV, MICHAEL BENZAQUEN, SEBASTIEN MICHELIN, LadHyX, UMR CNRS 7646, Ecole polytechnique, 91128 Palaiseau, France — Active droplets undergoing gradual micellar dissolution and spontaneous self-propulsion have recently received much interest as prototypical experimental realisations of synthetic micro-swimmers. While the self-propulsion of a single droplet has been widely studied and is known to arise above a critical advection-to-diffusion ratio, interactions and motion in complex environments remain mostly unexplored due to the non-linearity of the transport equation and its coupling with the flow that prevent the use classical superposition methods. Using a novel numerical framework relying on bi-spherical coordinate we solve the nonlinearly coupled hydrodynamics and solve dynamics exactly, enabling to characterise the rebound for various advection-to-diffusion ratios and to unravel the dominant interactions.

1This project has received funding from the European Research Council (ERC) under the European Unions Horizon 2020 research and innovation programme under Grant Agreement 714027 (SM).

Monday, November 25, 2019 5:16PM - 7:00PM – Session P36 Microscale Flows: Moving Contact Line and Thin Film Evaporation 618

- Serafim Kalliadasis, Imperial College London
designs of hierarchical surfaces for CHF enhancement. The detailed understanding of dry spot growth dynamics sheds light on more effective topography of structured surfaces plays a significant role in increasing the critical heat flux (CHF) during thin film evaporation by enhancing capillary-assisted liquid delivery to the evaporating thin film region. As the CHF is reached, evaporation becomes dominant, leading to the formation of dry spots. In this study, the contact line dynamics during dry spot growth is investigated for thin film evaporation on hierarchical micro/nanostructured surfaces with ZnO nanorods grown on silicon micropillars of varying spacings and heights. Using laser reflection interference microscopy, the 3D meniscus shape at the micropillar level and the contact line dynamics at two length scales are directly captured. Nanoscale receding front is found ahead of bulk receding during dry spot growth on hierarchical surfaces, where the bulk receding front follows a two-stage motion, slower around the micropillars and faster in-between pillars. This nanoscale precursor film, due to the presence of nanorods, contributes significantly to the evaporative heat flux. We fill this gap with a series of experiments in capillary tubes, and analyze theoretically the striking deviation from the Cox-Voinov relation. Finally, we point out several practical applications, from spin-coating of capillary tubes to fluid displacement in porous media.

The current view of dynamic contact angle is encapsulated in the seminal experiments of Harkins (1911) and Washburn (1921). By varying the liquid’s viscosity and injection rate, he determined a relation between the dynamic contact angle and the static contact angle, now known as Cox-Voinov relation. Very little is known, however, about the dynamics of the contact line in the reverse scenario: when a more viscous liquid is displaced by a less viscous fluid. Using laser reflection interference microscopy, the 3D meniscus shape at the micropillar level and the contact line dynamics at two length scales are directly captured. Nanoscale receding front is found ahead of bulk receding during dry spot growth on hierarchical surfaces, where the bulk receding front follows a two-stage motion, slower around the micropillars and faster in-between pillars. This nanoscale precursor film, due to the presence of nanorods, contributes significantly to the evaporative heat flux. The detailed understanding of dry spot growth dynamics sheds light on more effective designs of hierarchical surfaces for CHF enhancement.

We fabricated a humidity responsive film comprising a bundle of titanium oxide tubes that changes film’s curvature corresponding to the relative humidity. The mechanism of the change in curvature of the film can be explained by adsorption, condensation and evaporation of water molecules within the surface of the film. During adsorption, a liquid bridge forms because of growth of a water layer between tubes, thus contracting the gap between tubes. When vapor pressure exceeds the equilibrium vapor pressure, condensation occurs at the meniscus of the liquid bridge, thus expanding the gap between tubes. We obtained the adsorption and desorption isotherms for the humidity responsive film by measuring physical adsorption. Additionally, we demonstrated varying motions of the humidity responsive film when a water droplet was applied on the surface. The film could distinguish between saturated and oversaturated humidity conditions, such as fog and rain, respectively. Therefore, this humidity responsive film can be applied to environmental monitoring systems and possibly even to energy harvesting systems.

We fabricated a humidity responsive film comprising a bundle of titanium oxide tubes that changes film’s curvature corresponding to the relative humidity. The mechanism of the change in curvature of the film can be explained by adsorption, condensation and evaporation of water molecules within the surface of the film. During adsorption, a liquid bridge forms because of growth of a water layer between tubes, thus contracting the gap between tubes. When vapor pressure exceeds the equilibrium vapor pressure, condensation occurs at the meniscus of the liquid bridge, thus expanding the gap between tubes. We obtained the adsorption and desorption isotherms for the humidity responsive film by measuring physical adsorption. Additionally, we demonstrated varying motions of the humidity responsive film when a water droplet was applied on the surface. The film could distinguish between saturated and oversaturated humidity conditions, such as fog and rain, respectively. Therefore, this humidity responsive film can be applied to environmental monitoring systems and possibly even to energy harvesting systems.

We fabricated a humidity responsive film comprising a bundle of titanium oxide tubes that changes film’s curvature corresponding to the relative humidity. The mechanism of the change in curvature of the film can be explained by adsorption, condensation and evaporation of water molecules within the surface of the film. During adsorption, a liquid bridge forms because of growth of a water layer between tubes, thus contracting the gap between tubes. When vapor pressure exceeds the equilibrium vapor pressure, condensation occurs at the meniscus of the liquid bridge, thus expanding the gap between tubes. We obtained the adsorption and desorption isotherms for the humidity responsive film by measuring physical adsorption. Additionally, we demonstrated varying motions of the humidity responsive film when a water droplet was applied on the surface. The film could distinguish between saturated and oversaturated humidity conditions, such as fog and rain, respectively. Therefore, this humidity responsive film can be applied to environmental monitoring systems and possibly even to energy harvesting systems.
6:34PM P36.00007 Transitions between motion regimes of the three phase contact line during the pattern deposition of polymer from a volatile solution

Mohammad Abo Jabal, Anna Zigelman, Technion - Israel Institute of Technology — We investigate the deposition of polymer from a volatile solution. The interplay between different transport mechanisms in the volatile solution determines the motion regime of the three phase contact line and hence the morphology of the deposit. We observe monotonic slip, stick-slip, and periodic wetting-dewetting motions of the contact line. The deposits take the form of continuous coating in the former case and parallel stripes in the two later cases. To investigate transitions between the different motion regimes, we evaporate solutions of Poly-methyl-methacrylate and Poly-dimethyl-siloxane in toluene. The transitions between particular motion regimes of the contact line are connected to two types of competitions between physical mechanisms. A transport competition between polymer diffusion and convection determines the distribution of polymer in the volatile meniscus and hence determines the distribution of spatial variations in the excess energy at the free surface of the solution. A competition between evaporative and surface energy stresses in the liquid meniscus determines the motion of the contact line. We report the temporal variations of the contact line position during each motion regime and use theory to validate our experimental findings.

6:47PM P36.00008 The resolution of the moving contact line problem

Serafim Kalliadasis, Department of Chemical Engineering, Imperial College — At the heart of the problem is its multiscale nature: a nanoscale region close to the solid boundary where the continuum hypothesis breaks down must be resolved before phenomenological macroscale parameters such as contact line friction and slip, often adopted to alleviate the singularity, can be obtained. Here we will review recent progress made by our group to rigorously analyse the moving contact line problem and related physics from the nano- to macroscopic lengthscales. Specifically, to capture nanoscale properties and to establish a link to the macroscale behaviour, we employ elements from the statistical mechanics of classical fluids, namely density-functional theory (DFT). We formulate a new and general dynamic DFT (DDFT) which is coupled to hydrodynamics and we refer to as “hydrodynamic DDFT”. It is inherently multiscale bridging the micro- to the macroscale and retaining the relevant fundamental microscopic information (fluid temperature, fluid-fluid and wall-fluid interactions) at the macroscale level. Work analysing the contact line in both equilibrium and dynamics will be presented. To benchmark existing phenomenological models and reproduce some of their key ingredients. But its multiscale nature also enables us to unravel the underlying physics of the moving contact line, not possible with any of the previous approaches, and indeed show that the physics is much more intricate than the previous models suggest.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P37 Particle Laden Flows: Turbulence Modulation

5:16PM P37.00001 A new timescale for turbulence modulation by particles

Izumi Saito, Takeshi Watanabe, Toshiyuki Gotoh, Nagoya Institute of Technology — A new timescale for turbulence modulation by particles is introduced. This timescale is inversely proportional to the number density and the radius of particles and can be regarded as a counterpart of the phase relaxation time, an important timescale in cloud physics, which characterizes the interaction between turbulence and cloud droplets by condensation-evaporation. Scaling analysis and direct numerical simulations of dilute inertial particles in homogeneous turbulence suggest that turbulence modulation by particles can be expressed as a function of the mass-loading parameter and the Damkohler number, which is defined as the ratio of the turbulence large-eddy turnover time to the new timescale.

5:29PM P37.00002 Particle shape and orientation impact on the modulation of isotropic turbulence by Kolmogorov-scale size particles

Lennart Schneiders, California Institute of Technology, Konstantin Fröhlich, Institute of Aerodynamics of RWTH Aachen University, Wolfgang Schröder, Institute of Aerodynamics of RWTH Aachen University, JARA Center for Simulation and Data Science — The modulation of decaying isotropic turbulence by non-spherical particles of Kolmogorov-scale size is investigated via direct particle-fluid simulations. A Cartesian cut-cell method with dynamic mesh refinement and dynamic load balancing is applied to explicitly resolve the stresses acting on the fluid-particle interfaces. Up to 60,000 ellipsoids are fully resolved, requiring O(10^11) mesh points. The decay rates of the fluid and particle kinetic energy are found to increase with the particle aspect ratio. This is due to the particle-induced dissipation rate and the direct transfer of kinetic energy, both of which can be substantially larger than for spherical particles depending on the particle orientation. The extra dissipation rate resulting from the translational and rotational particle motion is quantified using a recently derived analytical model. This generic expression describes the impact of individual inertial particles on the local energy balance independent of the particle shape and enables to quantify the share of the rotational particle motion in the kinetic energy budget.

5:42PM P37.00003 Experimental Estimation of Turbulence Modification by Inertial Particles at Moderate $Re_\lambda$

Martin Obligado, Legi, Daniel Odens Mora, Legi / University of Washington, Alain Cartellier, Legi — Several experimental and numerical studies have aimed at quantifying the impact of inertial particles on turbulent kinetic energy, and turbulent kinetic energy dissipation ($\epsilon_p$) of particle-laden flows. We propose a new experimental method to estimate the carrier-flow dissipation $\epsilon_p$ in the presence of inertial sub-kolmogorov particles at moderate $Re_\lambda$ (Mora et al. PRF, 2019). Its foundations rely on the unladen flow dissipation calculation using the Rice theorem, and the density of zero crossings $n_\phi$ of the longitudinal velocity fluctuation coming from a laser doppler anemometry device. We show that, under some mild assumptions, that $\epsilon_p$ can be deduced from the value of $n_\phi$. Our experimental results provide strong evidence, regarding the non-negligible effect that dense sub-kolmogorov particles have on the carrier-flow energy cascade at $\phi_i = O(10^{-5})$, and $Re_\lambda \in [200 - 600]$. Our observations are consistent with previous two-way coupling DNS studies at similar concentrations. Our results may also have an impact on distinct phenomena on particle-laden flows that depend on the coupling of the particles with the flow, such as preferential concentration and settling velocity modifications.
5:55PM P37.00004 Drag and turbulence modulation by particles and viscoelasticity in channel flow\textsuperscript{1}, AMIR ESTEGHAMATIAN, TAMER ZAKI, Johns Hopkins University — Direct numerical simulations are performed to examine the effect of neutrally-buoyant, concentric channel flow. Comparison is drawn between the single-phase conditions and semi-dilute suspensions of particles (20% solid volume fraction), at various Weissenberg numbers. Viscoelastic effects are included using the FENE-P model, and an immersed-boundary method that is tailored for simulations of non-Newtonian particle-laden flows. Comparison is made between the single-phase and particle-laden conditions. Unlike the single-phase cases, the particle-laden flows show a drag enhancement with increasing viscoelasticity above a certain Weissenberg number. This effect is due to a drastic increase in polymer stresses near the surfaces of particles. Nonetheless, Reynolds stresses are effectively diminished in the particle-laden viscoelastic cases. In fact, the onset of viscoelastic drag enhancement in the particle-laden flows occurs when the Reynolds stresses are completely eradicated and the higher viscoelasticity results in increased polymer stresses.

\textsuperscript{1}This work was supported in part by the National Science Foundation.

6:08PM P37.00005 Stability of a Lamb-Oseen vortex in two-way coupled particle-laden flows, SHAULI SHUAJ, M. HOUSSSEM KASBAOUI, School for Engineering of Matter, Transport and Energy, Arizona State University — We investigate the stability of a columnar vortex tube loaded with inertial particles in Eulerian-Lagrangian simulations. The vortex tube is represented as a Lamb-Oseen vortex with a circulation-based Reynolds number equal to $Re_p = 50$. The particles considered have a small, yet non-zero, circulation Stokes number $St = 0.01$, and are dispersed at a volume fraction $10^{-3}$. We show that when the particle feedback on the fluid is neglected, the flow is stable to infinitesimal perturbations. However, when the two-way coupling is in account, a centrifugal Rayleigh-Taylor instability may develop leading to the destruction of the vortex structure.

6:21PM P37.00006 Modulation of coherent structures by inertial particles in a turbulent channel flow, HIMANSHU DAVE, MOHAMED KASBAOUI, Arizona State University — In this study, we explore how skin friction drag can be reduced by carefully modulating the near-wall coherent structures using inertial point particles. The particles in this study have a diameter $d_p = 0.2$ in wall units and are smaller than the smallest turbulent eddies. Owing to the preferential concentration mechanism, these particles are able to modulate the coherent structures as they get expelled from the vortical regions and gather in the extensional regions of the flow. In doing so, these particles may remove momentum from the energetic core of hairpin vortices while damping fluctuations in the sweep and ejection regions. This mechanism is investigated in Euler-Lagrangian simulations of a two-way coupled turbulent channel flow at a friction Reynolds number $Re_f = 180$. The particles are inertial, with a Stokes number $St^i = 1$, and present at a semi-dilute concentration, i.e., such that the average volume fraction is low ($\langle \phi \rangle = 2.23 \times 10^{-4}$) but the mass loading is significant ($M = 0.1$). We explore how particles distribute in the near-wall region, and how they modulate the turbulent structures in the fluid.

6:34PM P37.00007 Influence of sand particle on the wake of circular cylinder\textsuperscript{1}, GUOHUA WANG, Lanzhou University, DEPARTMENT OF MECHANICS, LANZHOU UNIVERSITY TEAM — We conducted two-phase flow around a circular cylinder experiment in wind tunnel and measured air and sand particle velocities synchronously in the sand-laden flow by Particle Image Velocimetry (PIV) technique. The influence of moving sand particle (with an average diameter of $159 \mu m$) on the wake of the circular cylinder was investigated. The results show that the wake of the circular cylinder inclines downward in the sand-laden flow, and the inclination angle increases with the increase of sand concentration. Under the influence of sand particles, the effect of mean streamwise velocity in the wake decreases with the increase of particle volume fraction. The settling particles cause the air in the wake flows downwards where the mean vertical velocity is no longer equal to 0. The streamwise and vertical turbulence intensities in the wake are weakened, in which the streamwise turbulence intensity in the wake gradually changes from a bimodal to an unimodal distribution with the increase of particle volume fraction.

\textsuperscript{1}This research was supported by the grants of the National Natural Science Foundation of China (nos. 11490553, 11702122).

6:47PM P37.00008 Scale-by-scale measurements of turbulence modification by heavy particles in homogeneous turbulence\textsuperscript{1}, ROUMAISSA HASSAINI, FILIPPO COLETTI, University of Minnesota — There is substantial evidence that, even at moderate concentrations, particles can significantly alter the turbulent fluctuations, but it is still debated under which conditions these will be excited or inhibited. The issue is complicated by the multiplicity of the influencing parameters, the scarcity of systematic experimental studies, and the difficulty in measuring fluid velocity in a particle-laden flow. We target this question using a facility featuring hundreds of individually controlled jets, in which homogeneous air turbulence with negligible shear and mean flow is generated and laden with microscopic solid particles. We vary the volume fraction by two orders of magnitude and use high-speed laser imaging at multiple resolutions. We combine particle tracking velocimetry (PTV) and particle image velocimetry (PIV) to simultaneously measure the position and trajectory of particles as well as the fluid velocity down to the Kolmogorov scales. We demonstrate the impact of the particles on the turbulent kinetic energy, dissipation rate, and energy spectrum, to an increasing extent with increasing volume fraction. In the considered range of parameters, gravitational settling is found to be a deciding factor as for whether turbulence is increased or reduced at different scales.

\textsuperscript{1}This project is funded by the US Army Research Office.

7:00PM P37.00009 Turbulence collapses at a threshold particle loading in a dilute particle-gas suspension, VISWANATH KUMARAN, Indian Institute of Science, PRADEEP MURAMALLA, Indian Institute of Technology Bombay, ANKIT TYAGI, Indian Institute of Science, PARTHA GOSWAMI, Indian Institute of Technology Bombay, INDIAN INSTITUTE OF SCIENCE COLLABORATION, INDIAN INSTITUTE OF TECHNOLOGY BOMBAY COLLABORATION — In order to examine the turbulence attenuation mechanism in a dilute particle-gas suspension, Direct Numerical Simulations (DNS) of a particle-gas suspension are carried out at a Reynolds number of about 3300 based on the average gas velocity and channel width. The particle Reynolds number based on the particle diameter and the flow velocity is about 42 and the Stokes number is in the range 9.5 – 377. The particle volume fraction is in the range $0 – 3 \times 10^{-3}$, and the particle mass loading is in the range 0 – 12. As the volume fraction is increased, a discontinuous reduction in the turbulent velocity fluctuations is observed at a critical volume fraction when the volume fraction is increased by $10^{-1}$. There is a reduction, by one order of magnitude, in the mean square fluctuating velocities in all directions, in the Reynolds stress and the turbulent energy production rate. Turbulence attenuation is due to a disruption of the turbulence production mechanism, and not due to the increased dissipation due to the particles. The turbulence collapse phenomenon is universal and is observed for different particle Reynolds numbers and for different models for the drag and lift force models, though the critical volume fraction depends on drag law and force model.
Long cross-over dynamics in capillary imbibition. Rodrigo Ledesma-Aguilar, Elfege Ruiz-Gutiérrez, Steven Armstrong, Gary G Wells, Simon Leveque, Celestín Michel, Smart Materials and Surfaces Laboratory, Northumbria University, Ignacio Pagonabarra, Áura Hernández-Machado, Department of Condensed Matter Physics, University of Barcelona — We present new experimental and theoretical results of the spontaneous capillary invasion of dry capillary tubes by viscous liquids. We show how deviations from Washburn’s law, which predicts a diffusive-like growth of the advancing meniscus, persist for much of the invasion process if not all. We identify two sources of hydrodynamic resistance that account for this effect: the difference in velocity between the fluid reservoir and the fluid within the capillary, and the motion of the meniscus itself. Both contributions give rise to power-law terms in the force balance, which introduce a long cross-over from the initial acceleration of the liquid to the asymptotic limit of Washburn’s law. Such long cross-over dynamics, which we call slowly-slowing-down dynamics, is likely to govern other systems in spontaneous capillary flow where a persistent resistance to growth is present.


Partial saturation in capillary rise into deformable porous material. Javed Siddique, Penn State York, Daniel Anderson, George Mason University — We investigate the partial saturation dynamics of capillary rise into a deformable porous material. Experiments show a deviation from the classical Washburn model dynamics after early times and our aim in this work is to investigate this deviation. We consider a three-phase mixture theory model with solid, liquid, and gas phases. We compare the results of our model to experimental data.

Imbibition with solidification in alumina feeding. Attila Kovacs, James Oliver, Chris Breward, Andreas Muench, University of Oxford — Liquid aluminium is produced from alumina by electrolysis in a Hall-Héroult cell. The process by which the alumina is fed into the cell influences the overall behaviour and efficiency of the cell. We develop a one-dimensional Stefan-type model for the imbibition of molten cryolite into a cold porous lump of alumina. In the small overheat limit, we analyse the small-time behaviour using the method of matched asymptotic expansions and find there to be locally self-similar solutions that describe analytically the competition between imbibition and freezing. Depending on the balance between these effects, the problem may exhibit two self-similar solutions (one being stable and the other unstable) or such a solution may cease to exist (so that imbibition is not possible without the molten cryolite freezing first on the exterior of the porous lump). Our asymptotic predictions are validated by direct numerical simulations that are also used to investigate the late time behaviour. In particular we predict the depth of imbibition before the cryolite freezes.

Extracting permeability model parameters for skin tissue from injection experiments. Pranav Shrestha, Boris Stoebier, The University of British Columbia — Hollow microneedles are medical devices used to inject fluid, such as vaccines, into the skin. As the fluid flows into the skin, a soft porous medium, it deforms the porous matrix. The fluid flow and solid deformation are coupled — the flow-induced deformation changes the porosity and permeability of the tissue, which in turn affects fluid flow. In our experiments, we injected water into excised porcine skin, while recording fluid flow-rate and visualizing the tissue cross-section in real time using optical coherence tomography (OCT). We performed digital image correlation on the OCT images to generate strain maps for quantifying tissue deformation. We used a spherical model of tissue expansion and a two-parameter exponential relationship between permeability and volumetric strain in tissue. Applying Darcy’s law to the measured fluid flow-rate and the strain maps over time yields the two parameters through an optimization algorithm. The fluid flow estimated from the optimized permeability model matched closely with the recorded flow-rate. The permeability-strain relationship can help improve the efficiency of fluid injections into the skin.

Funding from the Collaborative Health Research Projects by NSERC and CIHR, the Canada Research Chairs program and the UBC 4YF program.
6:34PM P38.00007 Fluid-structure interactions in a soft-walled Hele-Shaw cell¹.
SATYAJIT PRAMANIK, JIAN HUI GUAN, CHRISTOPHER W. MACMINN, University of Oxford — The interaction of viscous and interfacial flows with soft materials has recently attracted substantial interest from a variety of different perspectives. Here, we study these interactions in the context of a model problem: Flow in a deformable Hele-Shaw cell, where one wall is rigid and the other is soft. Combining experiments with mathematical modelling, we consider the coupling of flow and deformation during (a) the initial injection of viscous fluid into the empty cell (filling the problem), (b) the subsequent steady state during continued injection of the same fluid (the steady state), and (c) the relaxation of the cell after injection is stopped (the relaxation problem). We then discuss the implications of these results for hydrodynamic instabilities such as viscous fingering.

¹We acknowledge financial supports from EPSRC EP/P009751/1 and ERC H2020 805469.

6:47PM P38.00008 Modification of water injection monitoring for evaluation of hydraulic fracturing efficiency¹.
TIEN N. PHAN, MORTEZA DEJAM, Department of Petroleum Engineering, University of Wyoming, 1000 E. University Avenue, Laramie, WY 82071-2000, USA, MOHAMMADREZA KAMYAB, Corva AI, LLC, 16285 Park Ten Pl, Ste 210, Houston, TX 77084, USA — Measuring hydraulic fracturing efficiency is not a trivial task in the absence of production data. The traditional diagnostic plots for Hall Integral (HI) and derivative Hall Integral (dHI) against cumulative water injection allow for interpretation of plugging and fracturing behavior in injection wells. In this study, we adapt this method by including instantaneous shut-in pressure and cumulative slurry volume to understand fracturing efficiency. Increasing fracturing efficiency occurs when dHI is decreasing relative to cumulative slurry volume. We compute the change of dHI for all stages of data. Features extracted from our diagnostic plots like intercept, slope, and integral of dHI are correlated with computed fracture volume and half-length from microseismic data obtained in one hydraulically fractured well in Midland basin. The interpretation from correlation matrix allows us to identify which features can be used as qualitative predictors of fracturing efficiency. Using full stages of data, our derived parameters provide the implication on which stages should generate greater fracture volumes along the lateral of one single horizontal well. This study can improve the decision-making process for on-site evaluation of stage design.

¹The financial support from Corva AI, LLC is gratefully appreciated.

7:00PM P38.00009 Pore-Scale Modeling of Clogging and Erosion in Porous Media
EMILY DE JONG, Department of Chemical and Biological Engineering, Princeton University, NAVID BIZMARK, Princeton Institute for the Science and Technology of Materials, Princeton University, SUJIT DATTA, Department of Chemical and Biological Engineering, Princeton University — The transport of colloidal particles in porous media plays a key role in applications ranging from groundwater remediation, water filtration, and enhanced oil recovery. However, this process is difficult to model due to diverse processes that may arise, including particle advection through the pore space by fluid flow, adsorption and deposition onto the solid matrix, and erosion or suspension. Here, we analyze a continuum model to describe these physics for a single straight pore in a computationally-efficient simulation. The model accounts for geometry, advection through the pore space by fluid flow, adsorption and deposition onto the solid matrix, and erosion or resuspension. Our work thus provides a tractable description of the pore-scale physics of colloidal particle transport that can be generalized to more complex porous medium geometries.

Monday, November 25, 2019 5:16PM - 7:13PM – Session P39 Geophysical Fluid Dynamics Stratified Turbulence 6a - Elizabeth Follett, Cardiff University

5:16PM P39.00001 Analysis of Mixing in Rotating Stratified Turbulence
MATTHEW KLEMA, Colorado State University, ANNICK POUQUET, University of Colorado, DUANE ROSENBERG, 1401 Bradley Dr. Boulder, CO 80305, KARAN VENAYAGAMOORTHY, Colorado State University — This research introduces a parametric framework for the evaluation of rotating stratified turbulence (RST). Four dominant flow regimes in RST are delineated using the turbulent Froude number, Fr = /N, and the turbulent Rossby number, Ro = /k where k is the turbulent kinetic energy, / is the dissipation rate of k, N is the buoyancy frequency and is the Coriolis frequency. These four regimes reflect the relative contributions of buoyancy and rotation to the characteristics of the flow. Direct numerical simulations (DNS) are used for evaluation of the framework and analyzing the effects of forced rotation on turbulent mixing. The intensity of turbulent mixing is analyzed using the buoyancy Reynolds, ReB = /N², and the ratio of turbulent to molecular diffusivities, = /N. Results from the simulations show that forced rotation does not impact the behavior of the turbulence or impact scaling relationships with the turbulent Froude number when compared to non-rotating stratified DNS data. Ratios of the buoyancy frequency and the Coriolis rotational frequency is also shown to not be a useful ratio for the classification of stratified flow.

5:29PM P39.00002 Upward and downward transfer of energy in rotating stratified flows
ERNESTO HORNE, MOHAMMED HUSSINE HAMEDE, JEAN-MARC CHOMAZ, PAUL BILLANT, LdHyX, CNRS, Ecole Polytechnique, F-91128 Palaiseau CEDEX, France, GEOPHYSICAL FLOWS TEAM — We investigate experimentally and numerically the evolution of forced turbulence in the stronglystratified regime and for rotation rates covering the regimes associated to a direct and an inverseenergy cascade. The experiments are performed by means of a rotating table with a diameter of 2m. The energy is injected by periodically creating columnar dipoles. Direct numerical simulations (DNS) complement the experiments in order to reach higher Reynolds numbers. The turbulenceexhibits pancake structures with a large horizontal scale and a vertical thickness increasing with the rotation rate. Forstrong enough rotating rates, horizontal scales larger than the forcing scales appear in the experiments consistently with the observation of an inverse cascade in the DNS. We have found that the critical Rossby number below which the inverse cascade appears depends on the horizontal Froude number Frh.

JONATHAN AURNOU, UCLA, SUSANNE HORN, Coventry University, KEITH JULIEN, CU Boulder — Dimensional analysis is employed here to provide free-fall scaling estimates for the convective heat and momentum transport in the limit of rapid rotation, and to relate these to scalings for non-rotating Rayleigh-Bénard convection (RBC) systems. Our analysis shows that the scalings for free-fall dominated heat (Nusselt number, Nu) and momentum transfer (Reynolds number, Re) of rapidly rotating convection differ from their non-rotating RBC counterparts by a factor of Rfg, where Rfg = \tau_R^1/\tau_R is the free-fall Rossby number defined as the ratio of the characteristic rotation time \tau_R and the buoyant free-fall time \tau_R. Since Rfg < 1 in the rapidly rotating limit, our predicted rapidly rotating, free-fall transport rates remain far below the associated rates in non-rotating systems.

¹Carried out via support from the NSF Geophysics Program
surrounded by slowly ascending flow. The formation and longevity of the rings depends on the Prandtl number $Pr$ for a range of Taylor and Rossby (or flux Rossby) numbers, eventually breaking down into a grid of cyclonic vortices with descending flow. The formation and breakdown of transient axisymmetric rings of up- and down-welling fluid in impulsively started rotating Rayleigh Benard convection. First observed in laboratory experiments with constant negative heat flux at the top boundary [1-3], these rings form during spin-up for a range of Taylor and Rossby (or flux Rossby) numbers, eventually breaking down into a grid of cyclonic vortices with descending flow surrounded by slowly ascending flow. The formation and longevity of the rings depends on the Prandtl number $Pr$, with no sustained rings forming for $Pr < Pr_c(Ta, Ro)$. Furthermore, in the rapidly rotating regime and for the Rayleigh numbers $O(10^5 \text{--} 10^6)$ considered here, we find that the temperature boundary conditions on the top and bottom surfaces influence the ring dynamics and their breakdown. With Dirichlet conditions, the rings are less stable than for corresponding Neumann conditions, breaking down into sheet-like structures instead of individual vortices. The distinction resides in the nature of the stability of the upper boundary layer. References: [1] Boubnov and Golitsyn, J. Fluid Mech. 167, 503 (1986) [2] Vorobieff and Ecke, Phys. Fluids 10, 2525 (1998) [3] Zhong, Patterson and Wettlaufer, Phys. Rev. Lett. 105, 044504 (2010)

6:08PM P39.00005 Transient Convective Spin-Up Dynamics, S. RAVICHANDRAN, JOHN S. WETTLAUFER, Nordita, KTH Royal Institute of Technology and Stockholm University, SE-106 91 Stockholm, Sweden — We study numerically the formation and breakdown of transient axisymmetric rings of up- and down-welling fluid in impulsively started rotating Rayleigh Benard convection. First observed in laboratory experiments with constant negative heat flux at the top boundary [1-3], these rings form during spin-up for a range of Taylor and Rossby (or flux Rossby) numbers, eventually breaking down into a grid of cyclonic vortices with descending flow surrounded by slowly ascending flow. The formation and longevity of the rings depends on the Prandtl number $Pr$, with no sustained rings forming for $Pr < Pr_c(Ta, Ro)$. Furthermore, in the rapidly rotating regime and for the Rayleigh numbers $O(10^5 \text{--} 10^6)$ considered here, we find that the temperature boundary conditions on the top and bottom surfaces influence the ring dynamics and their breakdown. With Dirichlet conditions, the rings are less stable than for corresponding Neumann conditions, breaking down into sheet-like structures instead of individual vortices. The distinction resides in the nature of the stability of the upper boundary layer. References: [1] Boubnov and Golitsyn, J. Fluid Mech. 167, 503 (1986) [2] Vorobieff and Ecke, Phys. Fluids 10, 2525 (1998) [3] Zhong, Patterson and Wettlaufer, Phys. Rev. Lett. 105, 044504 (2010)

6:34PM P39.00007 Instability driven relaxation of an anticyclone, EUKOK YIM, Laboratory of Fluid Mechanics and Instabilities, EPFL, Lausanne, CH-1015, PAUL BILLANT, LadHyX, Ecole polytechnique, Palaiseau, 91120, France, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities, EPFL, Lausanne, CH-1015 — We study the nonlinear evolution of the centrifugal instability appearing in a columnar anticyclone using a semi-linear approach to model the transient unsteady flow evolution in a self-consistent manner. For anticyclones in a homogeneous viscous flow, the fastest growing instability is without oscillation in time but with a finite axial wavenumber. Hence, the self-consistent model is developed around the spatially averaged time dependent meanflow and the fluctuation, which reduces the problem from 2D nonlinear to 1D semi-linear. The two linear meanflow and fluctuation equations are coupled via the Reynolds stress of the fluctuations. At a given rotation ratio between the vortex angular velocity and the background rotation, only the most linearly unstable mode is considered for Reynolds numbers $Re = 800$ and $2000$ defined with the maximum angular velocity and the radius of the vortex. For both values of $Re$, the model predicts well the nonlinear evolution of the meanflow and the fluctuation amplitude. Higher harmonics are non-negligible only at the highest value of $Re$. The results show that the angular momentum of the meanflow is homogenized to a stable state via the action of the Reynolds stresses of the fluctuation.

6:47PM P39.00008 resonant excitation of modes and triadic resonance in a precessing cube, KE WIU, BRUNO WELFERT, JUAN LOPEZ, Arizona State University — The flow response of a rotating fluid-filled cube under precessional forcing is investigated numerically. The simulations are conducted over a wide range of forcing frequencies at various background rotation rates and a fixed precession angle of one degree. The periodic response is comprised of two main components: resonant excitation of the inviscid inertial eigenmodes of the cube, and inertial wave beams. The resonantly excited modes preserve the same spatio-temporal symmetries as the precessional forcing. Moreover, a symmetry-breaking response is also observed, which is due to triadic resonance, and the modes involved are identified.

7:00PM P39.00009 Aspects of Energetics of Stratified Turbulent Wakes, NIDIA CRISTINA REYES GIL, Cornell University, KRISTOPHER ROWE, Argonne National Laboratory, PETER DIAMENTISSIS, Cornell University, GREG THOMSEN, Wandering Wakes Research — The study of the turbulent wake generated by a bluff body moving through a stably stratified fluid has important applications in physical oceanography and marine engineering. Significant progress has been made towards understanding the structure and dynamics of the turbulent wake core, as well as the internal gravity wave (IGW) radiation emitted by the wake. Analysis of terms in the wake kinetic energy (KE) budget has demonstrated that viscous dissipation is a stronger sink than IGW radiation before $Nt = 10$, while the latter dominates during the mid-to-late non-equilibrium (NEQ) regime. Nevertheless, these processes do not close the KE budget: the largest gap occurring early in the NEQ regime. For a series of implicit large eddy simulations — spanning body-based initial Reynolds number $Re = 5 \times 10^3$, $10^5$ and $4 \times 10^5$, and Froude number $Fr = 4$, 16 and 64 — we calculate the complete KE budget of a stratified turbulent wake. The relative importance of viscous dissipation, IGW radiation, buoyancy flux, and nonlinear transport as sinks for wake KE is analyzed for different stages of the wake life cycle. Subsequently, numerical dissipation will be quantified and its impact compared to the physical processes in the KE budget.

1This work was funded by ONR Grant N00014-19-1-2101

Monday, November 25, 2019 5:16PM - 6:47PM
Session P40 Turbulent Boundary Layers: Control and Perturbations I 6b - Jonathan Morrison, Imperial College London
5:16PM P40.00001 Experimental Studies of Streamwise Response of the Turbulent Boundary Layer to a Periodic Actuation, MITCHELL LOZIER, FLINT O. THOMAS, STANISLAV GORDEYEV, Department of Aerospace and Mechanical Engineering, University of Notre Dame — It has been established that the dynamics of large-scale structures (LSS) in turbulent boundary layers (TBL) and near-wall small-scale turbulence are correlated. In these studies, a plasma based active flow control device was placed at sixty percent of the boundary layer thickness to introduce periodic disturbances into the wake region of the turbulent boundary layer. The boundary layer Reynolds number was low enough, Reτ = 700, so no natural large-scale structure was present. Via actuation, a synthetic large-scale periodic shear-layer-like structure was introduced into the boundary layer, and the TBL response to this synthetic structure at various wall-normal and streamwise locations downstream of the actuator was studied using a single hot-wire. Due to the periodic nature of the forcing, a phase-locked triple Reynolds decomposition of velocity was used to analyze the data. The modal component of velocity corresponding to the actuation frequency and the residual turbulence levels are the parameters of interest in this study. The dynamics of the LSS and small-scale structures were quantified using several modulation coefficients that correlate changes in modal velocity and residual turbulence with respect to phase. These modulation coefficients show a strong positive correlation in the inner and log region of the boundary layer. By measuring these quantities at several streamwise locations, the evolution of the synthetic large-scale structure and its modulating effect on the near-wall turbulence can be described.

5:29PM P40.00002 Turbulent boundary layer drag recovery downstream of spanwise wall oscillation. CHRISTOPHER BRYSON, FAZLE HUSSAIN, Texas Tech University — Spanwise wall oscillation (SWO) has been well researched for producing significant drag reduction. While the dynamics within the control region has been studied, the flow recovery downstream of the control region is poorly understood. Direct numerical simulation of a TBL with SWO is performed for a range of control velocity oscillation amplitudes A∗ = {10, 20, 30} and oscillation periods T∗ = {50, 100, 200}. It has been found that stronger drag reduction within the control region correlates with faster recovery to higher saturation levels of drag, which are also higher than an uncontrolled TBL. Compared with the uncontrolled case, visualization of control cases reveal that streak spacing is higher in the saturation region with increased wall normal velocity gradients in between, thus increased drag. The flow near the wall accelerates during the recovery phase producing a mean wall normal velocity that pulls streak transient growth (STG) vortices closer to the wall. The vortices then strengthen the streaks which spawn additional streamwise λ2 vortices by STG, contributing to the increased drag. Further details in terms of the connection between vortical structures and wall shear stress will be discussed.

5:42PM P40.00003 On the structure of compliant wall deformation forced by a turbulent boundary layer1. JIN WANG, SUBHRA KOLEY, JOSEPH KATZ, Johns Hopkins University — Our previous (Zhang et al 2017) study examined the pressure-deformation correlations in a compliant wall turbulent channel flow with a stief wall and submicron deformations. Aiming to extend the scope to two-way coupling, where the deformation size is several wall units (λ∗), theoretical analysis is used for selecting a compliant material (PDMS + silicone gel) with Young’s modulus (0.158 MPa), thickness (5mm), and shear speed (7.85 m/s) comparable to the freestream velocity (U’ = 1.2-6 m/s). Time-resolved (2 kHz) Mach-Zehnder Interferometry is used for mapping the deformation, and 2D PIV for measuring the flow. The deformations increase from submicron at U’ = 1.2 m/s to well above 20 µm at 6 m/s. The primary mode is advected at 0.66λ∗ for all wavenumbers, but the peak wavenumber in both directions remains nearly constant. In addition, high-frequency low wavenumber lateral waves appearing at broad streaks dominate at low U’ but persist at high-speed. Comparisons of the measured frequency spectra to 1-D linear models (Chase 1991, Benshop et al. 2019) show a good agreement for advected modes, but not for the lateral ones. At high U’, the compliant wall causes a sharp decrease in mean velocity at y∗ =10δ∗, consistent with DNS results (Rosti and Brandt 2017).

1ONR

5:55PM P40.00004 Input-output analysis of a turbulent separation bubble with a time-periodic base flow1. ALBERTO PADOVAN, CLARENCE ROWLEY, Princeton University, WEN WU, CHARLES MENEVEAU, RAJAT MITTAL, Johns Hopkins University — Direct numerical simulations are performed for three-dimensional turbulent boundary layer flow at Reθ∗ = 490, in which a suction velocity profile is imposed at the top of the computational domain to induce separation at the bottom wall. We study the input-out characteristics of perturbations about two different base flows: a spanwise-averaged, time-averaged base flow and a spanwise-averaged, time-periodic base flow. The first approach leads to the well-known resolvent analysis, through which we compute the optimal forcing and response modes at a given frequency and spanwise wavenumber. The latter approach leads to the formulation of the harmonic transfer function, a linear operator that governs the dynamics of fluctuations about time-periodic base flows. Within this framework, perturbations at different temporal frequencies are coupled to one another through the base flow, and we can therefore study the cross-frequency and energy transfer mechanisms. For the harmonic transfer function, we compute the optimal global forcing and response modes, which are full spatio-temporal flow fields. The cross-frequency modes provide insight into the spatial patterns that arise from the scattering of perturbations from the base flow.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-17-1-0084.

6:08PM P40.00005 Turbulent boundary layer injected with low blowing ratio effusion film1. JEREMY BASLEY, KEVIN GOUDER, JONATHAN F MORRISON, Imperial College London — Effusion cooling of turbine blades is used in jet engines to alleviate the thermal and shear strain they sustain while beneath the hot flow exiting the combustion chamber. This study focuses on the mechanisms underlying the interactions between the film-injected momentum and the incoming high Reynolds number turbulent flow. A large-scale low-velocity experiment is carried out in the closed-loop 10x5 wind-tunnel facility. The scaled-up effusion device consists of a plenum located directly underneath a thick plate pierced along a staggered grid of inclined D = 16mm diameter holes with a pitch of 5D. This setup is placed in a turbulent boundary layer, tripped and developing over 15 m. A range of injected velocities with respect to free-stream velocity (blowing ratio) is investigated with time-resolved planar PIV, complemented with hot-wire anemometry profiles, and wall-pressure measurements. The resulting time-resolved and space-extended data sets explain the favourable outcome of low blowing ratios, for which the shear-driven mixing of the effusion film is limited to near-wall region of the boundary layer. Results also suggest the effusion film effectively restricts the penetration of fluid from the outer region into the near-wall region of the boundary layer.

1The work is funded by the EPSRC grant (EP/P000878/1)
6:21PM P40.00006 Classification of forcing conditions in pulsatile turbulent pipe flow using Reynolds shear stress co-spectra1, ZUIN CHENG, THOMAS O. JELLY, SIMON J. ILLINGWORTH, IVAN MARIUS, ANDREW S.H. OOI, The University of Melbourne — The turbulence dynamics of pulsatile pipe flow are investigated using data obtained from direct numerical simulations at a mean friction Reynolds number of 180, 270 and 360. The forcing conditions are achieved by applying a time-harmonic axial pressure gradient. This study directs attention towards the frequency response of single- and two-point turbulence statistics to systematic variations in the forcing frequency. We propose a classification of the applied forcing conditions based on the Reynolds shear stress frequency co-spectra and the applied forcing frequency. We perform simulations based on this classification to extend the physical understanding of the phase dependence of single- and two-point turbulence statistics under high, very-high and ultra-high forcing frequencies, focussed around the frequency response of turbulence dynamics in time (frequency) and space (wavenumber) domains. Results also reveal a decoupling behaviour when the frequency of the forcing is higher than the highest frequency (smallest time-scale) of the energy containing motions in the Reynolds shear stress co-spectra.

1Australian Research Council

6:34PM P40.00007 Manipulating Near-wall Turbulent Boundary Layer by Unsteady Air-films1, CONG WANG, MORTEZA GHARIB, Caltech — Previously we have demonstrated that wall-attached air-films can be sustained in turbulent boundary layer (TBL) and dynamically modulated by pressure wave. This technique is effective in manipulating the near-wall turbulent shear flow. Here we show that in the presence of modulated air-films, the phase-averaged streamwise velocity demonstrates a Stokes type oscillatory motion. The near-wall viscous shear stress $\left(\nu \partial u/\partial y\right)$ is suppressed and negative Reynolds shear stress $(-\nu \partial v/\partial x)$ can be generated in the vicinity of air-films. Through a quadrant analysis, we identify a potential mechanism for the generation of negative Reynolds shear stress.

1This work was supported by the Office of Naval Research under Grant No. N00014-15-1-2479.


5:16PM P41.00001 Fully implicit force splitting scheme to two-phase lattice Boltzmann equation in pressure-velocity formulation2, TAEHUN LEE, City College of New York — We present a numerical procedure for solving the lattice Boltzmann equation (LBE) in the pressure-velocity formulation with the low Mach number approximation. We propose a unique algorithm based on the Strang splitting procedure to solve LBE for immiscible incompressible flows. Our procedure includes leading order intermolecular forcing terms within the streaming step, while keeping high-order forcing terms within the collision step such that conservative moments do not change due to collision. By coupling the finite difference/element method with a stable time-stepping technique, our scheme can easily handle stiff source term or external force. We will show that the perfect shift implementation is recovered under unity CFL condition as a special case of the proposed approach. The force splitting scheme is implemented in a fully implicit manner and applied to a novel pressure-velocity formulation of LBE. We have observed that the pressure-velocity formulation offers better numerical stability at high Reynolds numbers and reduced interfacial thickness, and the implicit formulation eliminates pressure oscillations. With the hope that this technique can be used for applications in complex geometries, benchmark calculations are performed on both uniform and non-uniform meshes.

5:29PM P41.00002 Streaming Formulation in Volumetric Lattice Boltzmann Method and Its Improvement1, HUIDAN (WHITNEY) YU, XIAOYU ZHANG, Indiana University-Purdue University Indianapolis — Volumetric lattice Boltzmann method (LVBM) has been specifically developed to deal with complex flow domains with or without willfully moving boundaries (Yu, et al, PRE, 2014). In the VLBM, fluid particles are uniformly distributed in lattice cells, instead of the traditional lattice nodes. A unique parameter, which represents the ratio of solid volume over the cell volume, is introduced to distinguish three types of lattice cells: solid cell (pure solid occupation), fluid cell (pure fluid occupation), and boundary cell (partial solid and partial fluid). Through this parameter, a volumetric bounce-back mechanism is uniquely included in the formulation of streaming process. As a result, extra interpolation/extrapolation to deal with arbitrarily oriented boundaries are avoided. Such a formulation significantly eases the handling of complicated geometries and promotes the computational efficiency without compromise of accuracy. In this work, we present the original formulation of the streaming process in VLBM and its modification. Quantitative validations in 3-D pulsatile duct flows with circular and rectangular cross sections respectively demonstrate the reliability of VLBM for solving unsteady flows.

1The authors would like to acknowledge US National Science Foundation through grant #NSF-CBET 1803845 and the University Fellowship of IUPUI

5:42PM P41.00003 Mesoscale Modelling of Nano-Particle Growth Under Flow, ROHAN VERNEKAR, TIMM KRÜGER, School of Engineering, Institute for Multiscale ThermoFluids, University of Edinburgh — Nanoparticles have wide potential for future applications, from drug delivery to surface coatings. Control over (e.g. silica) nanoparticle morphology, size, porosity and dispersity is crucial for realisation of their use. The physics of growth of such particles under flow conditions is not well understood, thus making their industrial scale-up challenging.

We present a mesoscale lattice Boltzmann (LB) algorithm that models growth of nanoparticles under particle resolved flow conditions via chemical species deposition. The method combines fluid LB for hydrodynamics, advection-diffusion LB for species transport with resolved Newtonian nanoparticle dynamics and a novel mesoscale adsorption boundary condition for growth. The algorithm is benchmarked for various 2D cases and provides excellent results. Our method enables the study of flow effects on growth, morphology and size distribution of nanoparticle suspensions, and advances nanoscale particle synthesis modelling.


5:55PM P41.00004 Simulation of Scalar Transport in a Non-Reacting Turbulent Jet using the Lattice Boltzmann Method. CHONG WU, WAI LEE CHAN, Nanyang Technological University, Singapore — In this work, numerical simulations of a non-reacting turbulent jet was performed with a lattice Boltzmann (LB) solver that is based on the open-source Palabos framework. The computational domain consists of a square nozzle, from which a free jet of Reynolds number of 10,000 was injected into a three-dimensional, open quiescent space. A scalar distribution function was introduced to describe the mixture fraction, with its transport properties extracted from a flamelet library. Subgrid-scale turbulence was described by another independent distribution function that closes the Smagorinsky model. To this end, the LB simulations are being run to a statistically-steady state, from which results can be verified with the local scaling of turbulence theories. Meanwhile, the scalar mixing profile at different axial locations will be investigated as it is critical to the implementation of flamelet-type combustion model. In addition, computational performance and simulation results of the LB method will be compared against that of large-eddy simulations, focusing in particular on the scalability of LB method.

6:08PM P41.00005 Phase-field Modeling and Simulation of Surfactant-Laden Multi-phase Flows using a Central Moment Lattice Boltzmann Method. FARZANEH HAjabDOLLAHI, KANNAN PREMNATH, SÁMUEL WELCH, University of Colorado Denver — Surfactants modulate interfacial flows in numerous multiphase dispersed systems. We will present a robust computational technique based on unified cascaded lattice Boltzmann methods (LBM) for two-phase flows at high density ratios, and for the capturing of the interfacial motions and surfactant dynamics. The cascaded LBM for two-phase flows, which computes the pressure and velocity fields, is based on a discretized modified continuous Boltzmann equation, where the effect of collisions is modeled by relaxation of different central moments to their equilibria and includes source terms for surface tension effects. The interfacial dynamics is represented by a conservative Allen-Cahn equation which is solved by another cascaded LBM. The transport of the surfactant concentration field, based on a free-energy functional and accounts for the energetic preference of surfactants to get adsorbed on interfaces, is evolved via yet another cascaded LBM. The effect of surfactants, i.e., the lowering of the local surface tension and the generation of Marangoni stresses, are introduced via a nonlinear interface equation of state based on the Langmuir isotherm. We will demonstrate the potential of our proposed formulation for various surfactant-laden two-phase flow cases.

6:21PM P41.00006 An improved coupled Immersed-Boundary-Lattice-Boltzmann solver for the simulation of particulate flows. EMMANOULI FALAGKARI, PDRA, TIMM KRUEGER, Lecturer — Our present understanding of the fundamental physical mechanisms of particle-fluid interactions is far from complete. We focus on the accurate computation of the hydrodynamic forces and the no slip condition on the boundary using the lattice-Boltzmann method for the solution of the flow field and a multi-direct forcing (MDF) immersed-boundary method for the fluid-structure interaction. We found that certain MDF schemes can become unstable after a certain number of iterations. The source of the instability has been identified in the iterative computation of the boundary force. We propose an alternative iterative scheme that significantly enhances the numerical stability by allowing the boundary force computation to relax at a different rate. The numerical accuracy and stability of the proposed scheme is demonstrated by simulating flows laden with moving finite-size particles, including a particle in shear flow and the sedimentation of single spherical and nonspherical particles in a cavity, demonstrating the importance of the accurate boundary force computation on the particle motion and dynamics. Good agreement between the present results and other schemes is obtained.

6:34PM P41.00007 An IB-LBM for modeling heat transfer and bushfire. FANG-BAO TIAN, LI WANG, School of Engineering & IT, University of New South Wales, Australia. JASON JOHN SHARPLES, School of Science, University of New South Wales, Australia — Catastrophic bushfire has happened many times over the world in the last two decades, destroying many assets and multiple facilities. The most devastating bushfires normally involve wind and terrain interactions. In order to predict the spreading process of bushfire in various geographical and weather conditions, it is necessary to improve understanding of the physical mechanisms of bushfire-terrain wind interaction. In this work, an immersed boundary method is developed for the numerical simulation of bushfire spreading mechanisms over complex terrains under different wind conditions. In this method, the lattice-Boltzmann method is used for the fluid dynamics. Large eddy simulation and wall model have been incorporate to handle the turbulence. The heat transfer is solved by using a finite difference method. The complex terrain is modeled by using an immersed boundary method. The numerical solver is validated by several benchmark cases, including heat transfer around a cylinder in a uniform flow, flow around a sphere at Re=10,000, and bushfire burning over a lee slope which is a typical terrain configuration.

Monday, November 25, 2019 5:16PM - 7:13PM — Session P42 Boundary Layers: Rough Wall Effects

5:16PM P42.00001 Modeling Turbulent Rough-Wall Flow with Pseudo Body Forces. GILES BRETERON, JUNLIN YUAN, MOSTAPHA AghAEI JOYBARI, Michigan State University — In RANS representations of turbulent flow over rough walls, it is desirable to describe flow variables as their superficial averages, so that the flow can be solved on a smooth-wall grid. The superficially averaged ‘extra’ terms in the x-momentum equation which describe the local effect of roughness on the flow, and which can be resolved by DNS with an immersed boundary method to enforce the precise local roughness boundary conditions, are \( \langle f_p \rangle \) and \( \langle f_r \rangle \). They describe averages per unit volume of the pressure dispersion and viscous stresses on account of roughness elements. As such, they are pseudo body forces distributed in the y direction, from the roughs to crests of the roughness sublayer. The distributed body force model appears to offer a more fundamental way of modeling general surface roughness than other approaches. In this talk, we present results which illustrate the applicability of the model for different kinds of roughness in fully-developed and boundary-layer flows. We also present computed distributions of \( \langle f_p \rangle \) and \( \langle f_r \rangle \) for flows over surfaces with different roughness textures, and describe the practical implementation of such a model within a wall function, for high Reynolds-number flow.

6:08PM P43.00002 An immersed-boundary lattice Boltzmann method for modeling particle-laden flows and its extension to three-dimensional flows. XIAO WANG, Post-Doc research associate at the Institute for Multiscale Thermofluids - The University of Edinburgh — Our present understanding of the fundamental physical mechanisms of particle-fluid interactions is far from complete. We focus on the accurate computation of the hydrodynamic forces and the no slip condition on the boundary using the lattice-Boltzmann method for the solution of the flow field and a multi-direct forcing (MDF) immersed-boundary method for the fluid-structure interaction. We found that certain MDF schemes can become unstable after a certain number of iterations. The source of the instability has been identified in the iterative computation of the boundary force. We propose an alternative iterative scheme that significantly enhances the numerical stability by allowing the boundary force computation to relax at a different rate. The numerical accuracy and stability of the proposed scheme is demonstrated by simulating flows laden with moving finite-size particles, including a particle in shear flow and the sedimentation of single spherical and nonspherical particles in a cavity, demonstrating the importance of the accurate boundary force computation on the particle motion and dynamics. Good agreement between the present results and other schemes is obtained.

6:21PM P43.00003 An immersed-boundary lattice Boltzmann method for simulating unsteady particle-laden flows. CHONG WU, WAI LEE CHAN, Nanyang Technological University, Singapore — In this work, numerical simulations of a non-reacting turbulent jet was performed with a lattice Boltzmann (LB) solver that is based on the open-source Palabos framework. The computational domain consists of a square nozzle, from which a free jet of Reynolds number of 10,000 was injected into a three-dimensional, open quiescent space. A scalar distribution function was introduced to describe the mixture fraction, with its transport properties extracted from a flamelet library. Subgrid-scale turbulence was described by another independent distribution function that closes the Smagorinsky model. To this end, the LB simulations are being run to a statistically-steady state, from which results can be verified with the local scaling of turbulence theories. Meanwhile, the scalar mixing profile at different axial locations will be investigated as it is critical to the implementation of flamelet-type combustion model. In addition, computational performance and simulation results of the LB method will be compared against that of large-eddy simulations, focusing in particular on the scalability of LB method.

6:34PM P43.00004 Simulation of Scalar Transport in a Non-Reacting Turbulent Jet using the Lattice Boltzmann Method. CHONG WU, WAI LEE CHAN, Nanyang Technological University, Singapore — In this work, numerical simulations of a non-reacting turbulent jet was performed with a lattice Boltzmann (LB) solver that is based on the open-source Palabos framework. The computational domain consists of a square nozzle, from which a free jet of Reynolds number of 10,000 was injected into a three-dimensional, open quiescent space. A scalar distribution function was introduced to describe the mixture fraction, with its transport properties extracted from a flamelet library. Subgrid-scale turbulence was described by another independent distribution function that closes the Smagorinsky model. To this end, the LB simulations are being run to a statistically-steady state, from which results can be verified with the local scaling of turbulence theories. Meanwhile, the scalar mixing profile at different axial locations will be investigated as it is critical to the implementation of flamelet-type combustion model. In addition, computational performance and simulation results of the LB method will be compared against that of large-eddy simulations, focusing in particular on the scalability of LB method.

6:41PM P43.00005 Phase-field Modeling and Simulation of Surfactant-Laden Multi-phase Flows using a Central Moment Lattice Boltzmann Method. FARZANEH HAjabDOLLAHI, KANNAN PREMNATH, SÁMUEL WELCH, University of Colorado Denver — Surfactants modulate interfacial flows in numerous multiphase dispersed systems. We will present a robust computational technique based on unified cascaded lattice Boltzmann methods (LBM) for two-phase flows at high density ratios, and for the capturing of the interfacial motions and surfactant dynamics. The cascaded LBM for two-phase flows, which computes the pressure and velocity fields, is based on a discretized modified continuous Boltzmann equation, where the effect of collisions is modeled by relaxation of different central moments to their equilibria and includes source terms for surface tension effects. The interfacial dynamics is represented by a conservative Allen-Cahn equation which is solved by another cascaded LBM. The transport of the surfactant concentration field, based on a free-energy functional and accounts for the energetic preference of surfactants to get adsorbed on interfaces, is evolved via yet another cascaded LBM. The effect of surfactants, i.e., the lowering of the local surface tension and the generation of Marangoni stresses, are introduced via a nonlinear interface equation of state based on the Langmuir isotherm. We will demonstrate the potential of our proposed formulation for various surfactant-laden two-phase flow cases.

1US National Science Foundation (NSF) under Grant CBET-1705663

1Would like to acknowledge support from the European Research Council (ERC).

2Post-Doc research associate at the Institute for Multiscale Thermofluids - The University of Edinburgh

3Lecturer at the Institute for Multiscale Thermofluids - The University of Edinburgh
5:29PM P42.00002 Multi-parameter prediction of roughness function and drag profiles in turbulent channel flows over rough walls1, MOSTAFA AGHAEI JOYYBARI, GILES J. BRERETON, JUNLIN YUAN, Department of Mechanical Engineering, Michigan State University, East Lansing, MI 48824 — We report on numerical experiments in which DNS is carried out for turbulent fully-developed channel flows over rough walls, describing 42 different surfaces at two frictional Reynolds numbers of 500 and 1000. The surfaces differ in roughness area distribution, effective slope, average inclination, porosity and degree of randomness. The dependence of equivalent sand-grain-height $k_s$ on different texture parameters was explored by using dimensional analysis and a differential evolutionary algorithm to optimize the coefficients of a single low-order multidimensional fitting function for all surfaces. It shows a strong dependence of $k_s$ on mean roughness height, effective slope and skewness, and fits the $k_s$ data with an error of no more than 20% in any of the 42 surfaces, even though the actual value of $k_s^+$ varies from 20 to 300 in these simulations. This corresponds to less than 10% error in prediction of the roughness function $\Delta u^+(k_s^+)$ for $k_s^+ \geq 25$. The same procedure is then used to model the total drag term in the Navier-Stokes equations when the Reynolds stress is described by the $i^2$-f model of Durbin (1991). It indicates a strong dependence of the drag profile on the roughness area distribution and on the profiles of $uv^+$ and $v^2$.

1Financial support from ONR (N00014-17-1-2102) is acknowledged.

5:42PM P42.00003 Determination of roughness scales that accurately predict frictional drag1, KAREN FLACK, MICHAEL SCHULTZ, US Naval Academy — Significant progress has been made towards the understanding of rough-wall boundary layers and the subsequent drag penalty. Continued progress is promising since a larger range of parameter space can now be investigated. Recent advances in rapid prototyping techniques enables the generation of systematic variations of roughness scales and computationally efficient simulations with creative surface mapping techniques allows for experiments and computations to investigate similar complexity. While a universal drag prediction correlation is still elusive and may not be possible, predictive correlations for classes of surface roughness pertinent to engineering applications seem achievable. Three surface parameters based solely on surface statistics are showing promise in predictive correlations for a range of studies. These include a measure of surface elevation ($k_{rms}$, $k_s$, $k_v$) a slope parameter (ES, solidity) and the skewness of the pdf. Other candidate parameters that may be useful in a predictive correlation or a surface filter are the streamwise and spanwise correlation lengths.

1Work supported by the Office of Naval Research.

5:55PM P42.00004 Turbulent boundary layer perturbation by two wall-mounted cylinders arranged in tandem at various spacings and height ratios, ALI HAMED, ADAM PETERLEIN, Department of Mechanical Engineering, Union College — The perturbation of a turbulent boundary layer by two cylindrical roughness elements in close proximity was experimentally investigated using particle image velocimetry (PIV). The two cylinders were arranged in tandem with center-to-center streamwise spacings of 2d, 4d, and 6d, where d is the diameter of the cylinders. The downstream cylinder had a fixed height (approximately 20% of the boundary layer thickness); the height of the upstream cylinder was varied to achieve upstream to downstream cylinder height ratios of 1, 0.75, and 0.5. The flow measurements were made at $Re = 57000$ (based on the boundary layer thickness and freestream velocity) and included measurements over an isolated cylinder as a baseline case. The results highlight the effects of sheltering by an upstream cylinder on the wake of the downstream cylinder. Flow features in the wake, including the downwash, upwash, recirculation zone, velocity deficit, Reynolds shear stress, and turbulent kinetic energy (TKE), are highly dependent on the degree of sheltering which is governed by both the streamwise spacing and height ratio. Additionally, proper orthogonal decomposition (POD) and quadrant analysis were used to examine the changes to the turbulence structure as a function of both the spacing and height ratio.

6:08PM P42.00005 Random roughness effects on the near-wall flow in the transitionally rough regime1, RONG MA, KARIM ALAME, KRISHNAN MAHESH, University of Minnesota — Direct numerical simulation of turbulent channel flow over a random rough wall is performed at $Re_r = 100$ and 600. The rough surface corresponds to the experiments of Flack and Schultz (personal communication). The skin friction coefficient of the random-rough channel matches with the experimental results of Flack and Schultz. The roughness effects on the near-wall regions of mean velocity, Reynolds stresses, pressure fluctuations and streamwise mean momentum balance are investigated. The statistics of wall-shear stress fluctuations in the peak (above the mean height location) and valley (below the mean height location) regions are examined. The probability distribution function of wall-shear stress shows a better collapse after subtracting the mean and normalizing by the root-mean-squared value. The distribution tail is widened by the random roughness, implying that the probability of extreme events is increased. The probability of extreme events in the random-rough channel increases with increasing $Re_r$, in accordance with previous studies on smooth-wall flows.

1This work was supported by the United States Office of Naval Research (ONR) Grant N00014-17-1-2308 managed by Drs. J. Gorski, T. Fu and P. Chang. Computing resources were provided by the Minnesota Supercomputing Institute (MSI). We are grateful to Professor K. Flack at the United States Naval Academy for providing us with the scanned surface data used in the present work.

6:21PM P42.00006 Towards Modelling the Downstream Development of a Turbulent Boundary Layer Following a Rough-to-Smooth Step Change in Surface Condition1, MOGENG LI, University of Melbourne, CHARITHA M. DE SILVA, University of New South Wales, DANIEL CHUNG, University of Melbourne, DALE I. PULLIN, California Institute of Technology, IVAN MARUSIC, NICHOLAS HUTCHINS, University of Melbourne — In this study we examine the effect of both the friction Reynolds number $Re_f$ and the roughness Reynolds number $k_s^+$ on a turbulent boundary layer following a rough-to-smooth step change in surface condition along the flow direction. To investigate the effect of $Re_f$, a set of wind-tunnel experiments is conducted at $k_s^+ = 160$ while $Re_f$ is varied from 7100 to 21000. Similarly, to examine the dependence on $k_s^+$, a set of measurements is conducted at $Re_f = 14000$ with $k_s^+$ ranging from 110 to 230. Hot-wire profiles are obtained on a logarithmically spaced grid up to 120 boundary-layer thicknesses downstream of the step change, and the local wall-shear stress is measured directly using oil-film interferometry. Using these data, we propose a new model of the recovering mean velocity profile which accounts for the well-known non-equilibrium behaviour of the internal layer. This mean velocity distribution is then evolved downstream of the step change using the integrated streamwise momentum equation to achieve a full prediction of the mean flow recovery.

1The financial support of the Australian Research Council is gratefully acknowledged.
Impact of spanwise effective slope on near-wall turbulence. THOMAS JELLY, NICHOLAS HUTCHINS, University of Melbourne, ANGELA BUSSE, University of Glasgow — Whereas streamwise effective slope (ES) is widely accepted as a key scaling parameter in the context of rough-wall turbulent flows (Napoli et al., J. Fluid Mech., 613:385-394, 2008), spanwise ES has received far less attention. Here, the statistical response of near-wall turbulence to systematic changes in spanwise ES is investigated by performing direct numerical simulations of rough-walled turbulent channel flow at a friction Reynolds number of 395. For the seven irregular surfaces considered in this study, the spanwise ES ranges from 0.35 to 0.10. All surfaces were synthesised with a near-Gaussian height distribution, i.e. negligible skewness and excess kurtosis, and a streamwise ES equal to 0.35. This allows the current study to focus on the impact of spanwise ES, since skewness, kurtosis and streamwise ES have been all effectively been eliminated as parameters. Starting from a baseline isotropic case, the hydraulic and hydrodynamic properties of each surface are compared as a function of spanwise ES. Details related to the Hama roughness function, turbulence intensities and dispersive stresses will be presented and discussed. Preliminary results from a complementary experimental campaign will also be presented.

Direct numerical simulation for irregular roughness on a curved surface. HAOLIANG YU, UMBERTO CIRI, ARIF MALIK, STEFANO LEONARDI, The University of Texas at Dallas — Surface roughness is critical to aerodynamic performance. Ice accretion, insect contamination, dust accumulation, and surface erosion represent major sources affecting surface perturbations in atmospheric applications. In wind energy, for example, insect contamination on turbine surfaces can significantly increase skin drag and reduce power production. Although some experiments and numerical simulations have been carried out to study roughness effects, the contributions of different roughness parameters are not well understood, particularly when over curved surfaces. Commonly used engineering models for roughness penalty prediction also demand more insights and validation. Therefore, this study includes direct numerical simulations for irregular roughness over a curved (airfoil) surface. A precursor simulation is conducted to generate fully developed turbulent inflow. The blade geometry and roughness are defined by the ray triangle intersection test and modeled using immersed boundary method. Resulting flow fields are analyzed and compared for the various parametric cases.

The effect of surface-obstacle geometry on maximizing air exchange over rough surfaces for use in direct air carbon capture assemblies. SIMONE STEW-ART, PAOLO LUZZATTO-FEGIZ, University of California Santa Barbara — Climate mitigation scenarios from integrated assessment models show it will likely be necessary to remove previously-released atmospheric CO2. A proposed implementation of negative emission technologies is direct air capture (DAC), which consists of box-like absorbers, through which air is passed and scrubbed of CO2. While the objective has been to scale up DAC to large arrays, the flow over these arrays has never been examined. The details of this flow are crucial; it is necessary for the first row of absorbers, it must mix quickly with the surrounding atmosphere before encountering the next row, or performance will be seriously degraded. To improve CO2 extraction from DACs, approaches to maximize the mass exchanges are investigated; exploiting the concept that flow over the DAC array is equivalent to flow over obstacles that comprise a rough surface. We perform reduced-scale experiments using model arrays in a water channel to study how the array geometry influences mass transport. We find that critical roughness spacings needed to avoid the development of recirculation regions are dependent on the aspect ratio of the obstacles. Thin obstacles have smaller critical spacings and result in stronger mixing, which supports the design of DAC arrays that minimize cost and land usage.

Direct Numerical Simulation of Wind Wave Growth. JIARONG WU, LUC DEIKE, Mechanical and Aerospace Engineering, Princeton University — The growth of wind wave is still an open question as current models often diverge in both the mechanism postulated and the growth rate predicted. There have been an increasing number of numerical studies on this subject, with the development of various numerical methods and increase in computational power. The difficulty, however, arises from three aspects: the modeling of the turbulent air boundary layer; the response of water to the wind forcing; and the coupling between the two. Separate numerical studies have been done on the first two but are insufficient in revealing the whole picture of wind wave interaction. Using the open source solver Basilisk, which solves the full two-phase air water incompressible Navier-Stokes equations with adaptive grids, we perform fully coupled wind wave simulation to study a range of parameters that factor into the growth model, including the friction velocity of air, initial wave amplitude, surface tension and viscosity, and to improve the current models of wind wave growth.

High-resolution DNS of Breaking Waves. WOUTER MOSTERT, Mechanical and Aerospace Engineering, Princeton University, STEPHANE POPINET, Institut Jean Le Rond d’Alembert, CNRS UMR 7190, Sorbonne Université, LUC DEIKE, Mechanical and Aerospace Engineering, Princeton University — We present bubble and droplet size distributions resulting from breaking ocean waves in deep water, using high-resolution three-dimensional direct numerical simulation. We use the open-source Basilisk code to simulate the viscous Navier-Stokes equations in two phases with surface tension at effective resolutions of up to 40963. The interface is represented and advected with a momentum-conservative volume-of-fluid scheme. The high effective resolutions are made possible with an octree adaptive mesh refinement scheme which is robustly implemented in Basilisk. The wave is initialized in one wavelength with an unstable third-order Stokes formulation, which produces local conditions leading to a plunging breaker which entrains air and ejects spray, which are directly resolved by the mesh. Varying the Bond and Reynolds numbers, which control surface tension and viscosity relative to the gravitational and inertial effects respectively, we discuss issues such as bubble breakup in turbulent flow; dimensionality in the transition to turbulence; droplet production and breakup; and numerical grid convergence.

This work is supported by a grant from NSF Physical Oceanography to L.D. (Grant No. 1849762.) and the Cooperative Institute for Earth System Modeling between NOAA-GFDL and Princeton University.
8:11AM Q01.00003 Effect of chemical herders on wave breaking1. LAKSHMANA CHANDRALA, FRANZ O’MEALLY, JOSEPH KATZ, Department of Mechanical engineering, Johns Hopkins University — Chemical surface-active agents (oil herders) could be used to concentrate oil slicks to facilitate in-situ burning after an oil spill. The water-insoluble but oil-soluble surfactants in commercial oil herders accumulate on the air-water interface and might alter the wave breaking process. In this study, the characteristics of mechanically generated breaking waves of varying energies are visualized in clean seawater and water treated with a herder containing 65% Span-20 and 35% 2-ethyl butanol at a concentration of 5.5 ml/m². The experiments are performed in a 6x0.3x0.6 m transparent tank and the waves are generated by translating a paddle. Multiple high-speed cameras follow the evolution of both waves. For a plunging breaker in clean water, prior to impact, the wave front contains multiple ripples and small fingers. In contrast, in treated water the wave-front is smooth, resulting in entrainment of a larger volume of air, and deeper subsequent penetration of the bubble cloud. Conversely, for relatively weak spilling breakers, adding the surfactant delays the wave breaking, and dampens the formation of capillary waves on the wave crest. Once breaking occurs, visually, there is no significant difference in the appearance and penetration of the waves.

1GOMRI

8:24AM Q01.00004 Study of ocean wave field reconstruction and effect of control parameter selection. JIE WU, XUANTING HAO, LIAN SHEN, University of Minnesota, Twin Cities & St, Anthony Falls Laboratory — In this research, we study the reconstruction of ocean wave field by optimizing the initial wave field for wave-phase-resolved simulation to best fit measurement data with noise. In reality, the measured data often contains only part of the information needed to reconstruct the entire wave field or is contaminated by noises. We investigate the effect of different control parameter selections on the wave field reconstruction performance using synthetic wave data. The initial condition of the wave field is constructed from the JONSWAP spectrum and its evolution is simulated using the high-order spectral method. Random noises are added to the wave field at each observation time instant to serve as the noise. The result shows that by incorporating relationship between the free surface elevation and the free surface velocity potential, the error between the reconstructed wave field and the true wave field can be reduced appreciably. This study provides guidance on choosing appropriate control parameters to improve the reconstruction and prediction performance in ocean wave applications.

8:37AM Q01.00005 ABSTRACT WITHDRAWN

8:50AM Q01.00006 Hysteresis phenomena in gravity–capillary waves on deep water generated by a moving two-dimensional/three-dimensional air-blowing/air-suction forcing1. YEUNWOO CHO, BEOMCHAN PARK, Korea Advanced Institute of Science and Technology — Hysteresis phenomena in forced gravity–capillary waves on deep water where the minimum phase speed c_{min}=23cm/s are experimentally investigated. Four kinds of forcings are considered; 2-D/3-D air-blowing/air-suction forcings. For a still water initial condition, as the forcing speed increases from zero towards a certain target speed (U), there exists a certain critical speed (U_{crit}) at which the transition from linear to nonlinear states occurs. When U<U_{crit}, steady linear localized waves are observed (state I). When U_{crit}<U<U_{min}, steady nonlinear localized waves including steep gravity–capillary solitary waves are observed (state II). When U≈c_{min}, periodic shedding phenomena of nonlinear localized depressions are observed (state III). When U>c_{min}, steady linear non-local waves are observed (state IV). Next, with these state-I, III and IV waves as new initial conditions, as the forcing speed is decreased towards a certain target speed (U_{final}), a certain critical speed (U_{crit,2}) is identified at which the transition from nonlinear to linear states occurs. When U_{final}<U_{crit,2}, linear state-I waves are observed. These are hysteresis phenomena which show the dependence of a state on its history starting from different initial conditions.

1This work was supported by the National Research Foundation of Korea (NRF-2017R1D1A1B03028299).

9:03AM Q01.00007 Air-induced axisymmetric sloshing waves on a water surface1. UTKARSH JAIN, FRANCESCO VIOLA, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids group, University of Twente — Here we experimentally study the surface wave trains generated by an oscillating disk placed above a water basin. The harmonic vertical motion of the disk yields a radial, periodic flow of air in the thin gap between the disk and the water. Although the disk is never in contact with the water, the oscillating air pressure in the gap is sufficient to excite a system of standing and travelling capillary-gravity waves at the free-water surface. The dynamics of these waves are measured both inside and outside the disk’s projection over the free water surface using an in-house experimental method based on total internal reflection. This allows us to reconstruct the instantaneous free surface elevation in the whole basin. To rationalize our experimental observations, we analytically solve the air flow below the disk using the lubrication equations. The resulting oscillating pressure is then coupled to the water phase through the dynamic condition at the free water surface, which forces axisymmetric waves on the entire liquid surface, for which we solve by means of Hankel transforms. Theoretical calculations and experiments show qualitatively similar behavior, and the wavenumbers measured experimentally are well reproduced by theory over a large frequency range.

1We acknowledge funding from SLING (project number P14-10.1) which is (partly) financed by the Netherlands Organisation for Scientific Research (NWO)

9:16AM Q01.00008 Generation of Water Waves by Underwater Multi-Point Action1. LEONARDO GOMRI, JUAN F. MARÍN, ISIS VIVANCO, Departamento de Física, Universidad de Santiago de Chile, Santiago, Chile, BRUCE CARTWRIGHT2, The University of Newcastle, Callaghan, NSW 2308, Australia — Wave generation in channels is usually achieved through wavemakers (moving paddles) acting on the surface of water. Although practical for this purpose, wavemakers have issues: they perform poorly in the generation of long waves and create evanescent waves in their vicinity. In this talk, we introduce a framework for wave generation through the action of an underwater multi-point mechanism. We analyze the linear response of waves in a uniform channel in terms of the frequency and wavelength of the bottom action. The system behaves as a long-pass filter in space and a high-pass filter in time with a sharp resonance limited by viscosity. The framework naturally solves the problem of the performance for long waves and reduces evanescent waves to thin boundary layers at the bottom. We also show that a proper synchronization of an orbital motion on the bottom can produce waves that mimic deep water waves with great accuracy. This last feature has been proved to be useful and efficient to study fluid-structure interaction in simulations based on smoothed-particle hydrodynamics.

1Funded by Fondecyt 11170700, Australian Research Council-Linkage Project LP160100391 and USA1899-Vridei 041931YZ-PAP.
2Pacific Engineering Systems International, Glebe, NSW 2037, Australia
converging channel, on a suspended Silicone oil droplet (radius gradient in the bulk flow imposes viscous stresses on a deformable interface. Here we study the effect of a bulk flow of Castor oil, through a

**ARTIOM KOSTIOUK, AMIR RIAZ, University of Maryland, College Park** — In low Reynolds number multiphase flows, existence of a velocity profile, which agrees well with experimental determination of the flow velocity. Suspending Silicone oil droplets into the single phase flow profile, also on a scale comparable to the Faraday wavelength. This behavior conditions the droplet's cycle consists of periodic velocity oscillations, on a scale comparable to the Faraday wavelength. This behavior conditions the droplet's period is prolonged by the presence of additional surfactants but the onset of cooling is not significantly affected.

$^1$The support of the National Science Foundation is gratefully acknowledged.

**Tuesday, November 26, 2019 7:45AM - 9:42AM**

**Session Q02 Drops: General I**

**7:45AM Q02.00001 Velocity Oscillations of a Walking Droplet**$^1$. SAM TURTON, MATTHEW DUREY, JOHN BUSH, Massachusetts Institute of Technology — Couder et al. (2005) demonstrated that a droplet bouncing on the surface of a vertically oscillating bath may destabilize to a walking state characterized by rectilinear motion across the bath surface at a constant speed. When a walking droplet is perturbed, there is experimental (Wind-Willansen et al. (2013)) and theoretical (Bacot et al. (2019)) evidence suggesting that its velocity may return to its free walking speed via over- or underdamped oscillations. By revisiting the stroboscopic pilot-wave model of Oza et al. (2013), we demonstrate that linear stability analysis of the walking state predicts velocity oscillations over a lengthscale that becomes commensurate with the Faraday wavelength as the vibrational acceleration approaches the Faraday threshold. Furthermore, we demonstrate that this model predicts that the walking state destabilizes via a subcritical Hopf bifurcation, where the associated unstable limit cycle consists of periodic velocity oscillations, also on a scale comparable to the Faraday wavelength. This behavior conditions the droplet’s histogram to develop a coherent structure on the scale of the Faraday wavelength, and so provides new insight into the emergence of quantum-like statistics from pilot-wave hydrodynamics.

$^1$The authors acknowledge the financial support of the NSF through grants DMS-1614043 and CMMI-1727565.

**7:58AM Q02.00002 Experiments on Droplet Deformation and Lag in a Low Reynolds Number Flow**. DAVID BIGIO, ADITYA N. SANGLI, University of Maryland, College Park, MARGARET LO, Cornell University, ARTIOM KOSTIOUTK, AMIR RIAZ, University of Maryland, College Park — In low Reynolds number multiphase flows, existence of a velocity gradient in the bulk flow imposes viscous stresses on a deformable interface. Here we study the effect of a bulk flow of Castor oil, through a converging channel, on a suspended Silicone oil droplet (radius $\sim 2$ mm). We apply Lubrication theory to study the bulk flow and approximate the single phase flow profile, which agrees well with experimental determination of the flow velocity. Suspending Silicone oil droplets into the flow, we study the effect of a constant extensional rate on the droplet imposed by the bulk flow. The degree of droplet deformation depends on the initial Capillary number and heuristic deductions of the critical Capillary number are made. Experimental observations also indicate a lag in the droplet advection velocity compared to the bulk flow at the same point and we present a physical mechanism to explain the observation.

**8:11AM Q02.00003 Direct Numerical Simulations of Droplet Deformation in Low Reynolds Number Extended Flows**. ADITYA N. SANGLI, BERNARD CHANG, DAVID BIGIO, AMIR RIAZ, University of Maryland, College Park — We study the effect of stagnant and non-stagnant extensional flows on a suspended immiscible droplet using direct numerical simulations. We use projection method to solve the governing equations of motion, and the deformable interface is treated using level set method. This scheme is used to simulate the behavior of a droplet in a hyperbolic flow as a low Reynolds number numerical analogue of the classic four-roll mill experiments. Results indicate that the critical Capillary number of the droplet increases with reduction in Reynolds number and approaches results from the four-roll mill study which approximated a Stokesian flow. To study non-stagnant extensional flows, we simulate the motion of a Silicone oil droplet (radius $\sim 2$ mm) in a Castor oil matrix through a hyperbolic converging channel. Streamwise stretching of the droplet was observed and plots of droplet draw ratio vs time and position of the droplet in the flow are validated with experimental results.

**8:24AM Q02.00004 The resonance of water balloons and water drops**$^1$. CHUN-TI CHANG, National Taiwan University, PAUL STEEN, Cornell University — What happens if a sessile drop is covered with a membrane and mechanically oscillated? In this study, we examine the resonance behaviors of water balloons, and we compare them to those of drops. It turns out that balloons and drops exhibit a variety of similar dynamical behaviors: families of corresponding modes, similar scalings for onset thresholds and almost identical dispersion relations. However, balloons and drops may synchronize differently with the forcing. In this talk, we shall carefully examine different aspects of the resonance of balloons and drops. Ultimate, we shall identify the extent to which one can understand the dynamics of balloons in terms of that of drops.

$^1$Ministry of Science and Technology, Taiwan, Grant No. MOST 107-2218-E-002-070-MY3

**8:37AM Q02.00005 On the Basset-Boussinesq history force of a fluid sphere**. DOMINIQUE LEGENDRE, IMFT-Toulouse — We consider the Basset-Boussinesq (history) force experienced by a spherical drop. The kernel of the Basset-Boussinesq force has not been determined so far when internal circulation of the fluid occurs. We first characterize the slip at a fluid sphere interface. Under both steady and unsteady conditions, the corresponding slip length is remarkably uniform along the fluid sphere interface and is directly related to the viscosity ratio. Combining the analytical expression of the Basset-Boussinesq kernel obtained for a solid sphere with interface slip and the obtained description of the slip at the fluid-fluid interface, we were able to describe for the first time the Basset-Boussinesq history force acting on a spherical drop. This expression is valid whatever the viscosity ratio from bubbles to viscous drops.
8:50AM Q02.00006 Hydrodynamics of a hollow water droplet falling in air, MOONIKA BALLA, Indian Institute of Technology Hyderabad, India, MANOJ TRIPATHI, Indian Institute of Science Education and Research Bhopal, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We study the hydrodynamics of a compound/hollow water droplet falling in air by conducting three-dimensional numerical simulations of the Navier-Stokes and the continuity equations using a finite volume based volume-of-fluid method. The various physical parameters influencing the dynamics are the thickness of the hollow droplet, inertia and surface tension at the air-water interface. It is found that the droplet exhibits shape oscillations. The oscillations of the inner interface (inner air bubble) of the hollow droplet is periodic with a time period about half of that of a normal droplet. The deformation of the outer interface of the hollow droplet is aperiodic and breaks up when the deformation is more. Increasing the thickness of the hollow droplet decreases the oscillations of both the inner and outer interfaces. As expected, the oscillations decay with time at low inertia. As the thickness of the hollow droplet decreases a spike like structure appears at the bottom of the hollow droplet. This instability is enhanced as the surface tension and the ratio of the viscous force to the gravitational force are decreased. The velocity contours are used to explain the behaviour observed in the present study.

9:03AM Q02.00007 Force analysis acting on the moving droplet in air-flow1, YEWON KIM, HYUNGMIN PARK, Seoul National University, ALIDAD AMIRFAZLI, York University — Movement of a water droplet over a surface due to a shearing airflow was studied in a wind tunnel. Water droplets (5-20 μl) were placed on PET (hydrophilic) and PTFE (hydrophobic) surfaces; airflow velocities up to 25 m/s and accelerations between 4.4-10 m/s² were used. The Reynolds number is under 3,400 based on the free-stream air velocity (U∞) and the height of the droplet (h). High-speed cameras from top and side views were used to evaluate the droplet velocity, acceleration, and contact angle hysteresis. Droplets start to move at different air velocities depending on the droplet volume, flow acceleration, and surface wettability. Also, droplets show different acceleration patterns for different experimental case. For example, when flow acceleration is 4.4 m/s², 5μl droplet on the PTFE surface start to move at lower free-stream velocity with lower droplet acceleration compared to those on the PET surface. To understand the behavior of droplets, we constructed a model by considering the drag, adhesion, and viscous forces acting on the moving droplet. We would discuss the relation between the droplet behavior and airflow state (velocity and acceleration) and validate our force balance model for the droplet.

1Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02, IT12041 MITACS GRA-NRF) funded by Korea government, Korea Coast Guard, and MITACS.

9:16AM Q02.00008 Droplet condensation patterns: universality or non-universality?, LAURA STRICKER, ETH Zurich, JUERGEN VOLLMER, University of Leipzig, ROBERT STYLE, ERIC DUFRESNE, ETH Zurich — When a flux of supersaturated vapour gets into contact with a solid substrate, a condensation process can originate, leading to the formation of droplets patterns on the substrate (“breath figures”). The interest for breath figures is both practical and theoretical. We present here an experimental study of three-dimensional breath figures forming on a two-dimensional substrate. By making use of scaling concepts, we indentify a series of characteristic exponents, such as the so-called polydispersity exponent, characterizing the droplet size distribution. We proof the internal consistency of the results and we compare them to the predictions of the classical theory. According to such a theory, the polydispersity exponent should be a universal number, depending only on the dimensionality of the system. However, more recent theoretical studies claimed that the polydispersity exponent should heavily depend on the micrometric details of the condensation process (e.g. contact angle and critical nucleation radius of the droplets). We investigate the issue and we try to provide an answer, based on the experimental data.

9:29AM Q02.00009 Ultrasonically-Enhanced Condensation by Induced Interfacial Droplet Ejection, THOMAS BOZIUK, MARC SMITH, ARI GLEZER, Georgia Institute of Technology — Ultrasonic (MHz range) actuation at the liquid-vapor interface of vapor flowing over a slow moving sub-cooled liquid layer exploits the differences in acoustic impedance to form a train of droplet that are ejected into the vapor. The increased interfacial surface area of the ejected droplets result in increased heat transfer between the vapor and the liquid and a significant increase in condensation without inducing a significant increase in upstream pressure. The enlarged droplets drop back into the subcooled liquid stream under gravity. A two-stream liquid-vapor experimental setup was designed to assess the effects of the acoustic actuation on the formation and ejection of the subcooled liquid droplets and the enhanced condensation over several combinations of relative flow rates and liquid subcooling, the observable increases in the sensible heat of the liquid stream as well as increased rate of mass transfer from the vapor to liquid stream. The evolution of the droplet-laden vapor within a mixing volume is visualized using high-speed imaging and is used to investigate the effects of the residence time of the droplets within the vapor volume and possible applications to acoustically-driven heat exchangers.

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q03 Surface Tension III

7:45AM Q03.00001 Effect of surfactants on jet break-up and drop formation in inkjet printing1, EVANGELIA ANTONOPoulos, OLIVER G. HARLEN, MARK A. WALKLEY, NIKIL KAPUR, University of Leeds — A key challenge in developing new applications of inkjet technology is to produce inks that can be jetted to form individual droplets and to transport functional components needed for the application. The development of mathematical models that allow fluid jetting behaviour to be determined as a function of fluid properties would allow optimisation to be carried out in-silico before creating the inks and verifying the performance. Surfactants are often added to aqueous inks in order to modify the surface tension. However, the rapid expansion of the free surface during the fast jetting process means local areas of the surface will be depleted of surfactants leading to surface tension gradients. Using high speed video we will compare the jetting behaviour of fluids with and without surfactants in an industrial inkjet print-head. We also present numerical simulation of inkjet break-up and drop formation in the presence of surfactants investigating more closely both the surfactant transport on the interface and the influence of Marangoni forces on break-up dynamics.

1Supported by the EPSRC [grant EP/L01615X/1]
with surfactant in laminar three-dimensional flows.

11:07 AM Q03.00002 The role of surfactants on the interfacial stability of multilayer shear flows.

11:10 AM Q03.00003 Underwater acrobatics of partially-coated spheres.

11:13 AM Q03.00004 Dynamic Wetting Failure in Curtain Coating: Comparison of Model Predictions and Experimental Observations.

11:16 AM Q03.00005 an Armored Droplet Approaching a Fluid-Fluid Interface.

11:19 AM Q03.00006 A theory for the drag reduction of superhydropobic surfaces with surfactant in laminar three-dimensional flows.

11:22 AM Q03.00007 An asymptotic model for advanced design.

11:25 AM Q03.00008 The role of surfactants in interfacial instabilities.

11:28 AM Q03.00009 Micellar versus surfactant film models: Where do the two meet?

11:31 AM Q03.00010 Simulating surfactant dispersion in complex fluids.

11:34 AM Q03.00011 Modelling of surfactant transport in non-Newtonian fluids.

11:37 AM Q03.00012 An interfacial nanodroplet formation mechanism.

11:40 AM Q03.00013 An interfacial water demineralization mechanism.

11:43 AM Q03.00014 The effect of surfactants on the interfacial stability of multilayer shear flows.

11:46 AM Q03.00015 An experimental study of surfactant film dynamics.

11:49 AM Q03.00016 The role of surfactants in interfacial instabilities.

11:52 AM Q03.00017 An interfacial nanodroplet formation mechanism.

11:55 AM Q03.00018 An interfacial water demineralization mechanism.

11:58 AM Q03.00019 The effect of surfactants on the interfacial stability of multilayer shear flows.

12:01 AM Q03.00020 An experimental study of surfactant film dynamics.

12:04 AM Q03.00021 The role of surfactants in interfacial instabilities.

12:07 AM Q03.00022 An interfacial nanodroplet formation mechanism.

12:10 AM Q03.00023 An interfacial water demineralization mechanism.

12:13 AM Q03.00024 The effect of surfactants on the interfacial stability of multilayer shear flows.

12:16 AM Q03.00025 An experimental study of surfactant film dynamics.

12:19 AM Q03.00026 The role of surfactants in interfacial instabilities.

12:22 AM Q03.00027 An interfacial nanodroplet formation mechanism.

12:25 AM Q03.00028 An interfacial water demineralization mechanism.

12:28 AM Q03.00029 The effect of surfactants on the interfacial stability of multilayer shear flows.

12:31 AM Q03.00030 An experimental study of surfactant film dynamics.

12:34 AM Q03.00031 The role of surfactants in interfacial instabilities.

12:37 AM Q03.00032 An interfacial nanodroplet formation mechanism.

12:40 AM Q03.00033 An interfacial water demineralization mechanism.

12:43 AM Q03.00034 The effect of surfactants on the interfacial stability of multilayer shear flows.
9:03AM Q03.00007 Three-dimensional simulations of surfactant-contaminated flows in superhydrophobic microchannels. SCOTT SMITH, FERNANDO TEMPRANO-COLETO, UCSB, FRANCOIS PEAUDECERF, ETH Zurich, JULIEN LANDEI, U. Manchester, FREDERIC GIBOU, PAOLO LUZZATTO-FEGIZ, UCSB — Trace amounts of surfactants are now known to severely limit the drag reduction along superhydrophobic surfaces, due to Marangoni stresses, as demonstrated in Peaudecerf et al. (PNAS, 2017), and Song et al. (PRF, 2018). When driving flow in superhydrophobic microchannels, surfactants adsorb onto the air-water interface (known as the plastron) and cause adverse Marangoni stresses in the opposite direction to the driving flow. There is currently an effective model that considers two-dimensional surfactant-laden flow in superhydrophobic channels (Landel et al. arXiv:1904.01194, 2019). Here we perform three-dimensional simulations of the full problem using COMSOL, in order to test our extension of this model to account for the drag of a fully three-dimensional flow. We are able to compute drag reduction as a function of the microchannel geometry, surfactant concentration, and the characteristic dimensionless numbers of the momentum and surfactant transport equations. These simulation results are then used to explore physical trends and validate our extended model.

1Supported by ARO MURI W911NF-17-1-0306

9:16AM Q03.00008 Suspension coating on a fiber. EMILIE DRESSAIRE, BRIAN DINCIAU, UCSB, QUINTEN MAGDELAINE, SVI (CNR/SAINT GOBAIN), ETHAN MAI, UCSB, MARTIN BAZANT, MIT, ALBAN SAURET, UCSB — The thickness of the coating layer entrained by a solid withdrawn from a bath depends on the physical properties of the liquid, the withdrawal speed, but also on the substrate geometry. In particular, many common substrates that are subjected to liquid immersion and withdrawal have the general shape of a thin cylinder such as needles, wires, and fibers. We investigate glass fibers as a model substrate and demonstrate that their diameter plays a dominant role in the particle entrainment and coating by suspensions. We identify experimentally and rationalize different coating regimes of the fiber: at small capillary number, only a liquid film coats the fiber. At intermediate capillary numbers, a heterogeneous coating made of clusters of particles is observed. Finally, at large capillary number, the thickness of the entrained film is captured using the effective viscosity of the suspension. Our results demonstrate that varying the size of the fiber leads to a new degree of control in the entrainment of particles via capillary filtering.

9:29AM Q03.00009 Weakly-nonlinear evolution of surface-tension driven waves in the presence of viscosity. QUINTON FARR, ROUSLAN KRECHETNIKOV, University of Alberta — Weakly nonlinear models describing the evolution of water waves such as the Korteweg-de Vries and nonlinear Schrödinger equations have been widely studied because of their ability to capture the behavior of interesting physical phenomena through the reduction of a full problem to a much simpler system retaining only first order nonlinear effects. While a large class of these reductions for gravity-driven waves are well-understood, situations where surface tension is the only driving force require further attention. We consider wave motion on a one-dimensional thin liquid film of infinite extent and on the perimeter of a two-dimensional liquid drop, forced by surface tension in lieu of gravity. Viscosity is also included to understand the effect of dissipation. Our investigation leads to new equations governing surface tension-driven waves. We discuss their properties, special solutions, and physical implications.

Tuesday, November 26, 2019 7:45AM - 9:42AM
Session Q04 Suspension : Rheology 203 - Elisabeth Guazzelli, CNRS

7:45AM Q04.00001 Constitutive model for time-dependent flows of shear-thickening suspensions. JURRIAAN GILLISSEN, University College London, CHRISTOPHER NESS, JOSEPH PETERSON, University of Cambridge, HELEN WILSON, University College London, MICHAEL CATES, University of Cambridge — We develop a tensorial constitutive model for dense, shear-thickening particle suspensions subjected to time-dependent flow. Our model combines a recently proposed evolution equation for the suspension microstructure in rate-independent materials with ideas developed previously to explain the steady flow of shear-thickening ones, whereby friction proliferates among compressive contacts at large particle stresses. We apply our model to shear reversal, and find good qualitative agreement with particle-level, discrete-element simulations whose results we also present.

1We acknowledge financial support from the Engineering and Physical Sciences Research Council of the United Kingdom Grant No. EP/N024915/1, and from the European Research Council under the Horizon 2020 Programme, ERC grant agreement number 740269. MEC is funded by the Royal Society.

7:58AM Q04.00002 The influence of surface roughness on the rheology of immersed and dry frictional spheres. ELISABETH GUAZZELLI, MSc, CNRS, University of Paris, FRANCO TAPIA, OLIVIER POULIQUEN, Aix Marseille University, CNRS, IUSTI — Pressure-imposed rheometry is used to examine the influence of surface roughness on the rheology of immersed and dry frictional spheres in the dense regime. The quasi-static value of the effective friction coefficient is not significantly affected by particle roughness while the critical volume fraction at jamming decreases with increasing roughness. These values are found to be similar in immersed and dry conditions. Rescaling the volume fraction by the maximum volume fraction leads to collapses of rheological data on master curves. The asymptotic behaviors are examined close to the jamming transition.

8:11AM Q04.00003 Mechanism of contact network formation leading to discontinuous shear thickening in dense suspensions. PRABHU NOTT, TABISH KHAN, Indian Institute of Science — The phenomenon of discontinuous shear thickening (DST) of dense particle-liquid suspensions has received considerable attention in recent years, and it is now generally believed that the formation of a Coulomb friction-mediated particle contact network causes the dramatic rise in viscosity at a critical shear stress. A common feature of most experimental studies is that the suspension is ‘prepared’ by pre-shearing for a long time before conducting the shear rate or stress sweep. The implicit assumption in this protocol is that in the prepared state, the stress responds at a critical shear stress. A common feature of most experimental studies is that the suspension is ‘prepared’ by pre-shearing for a long time before conducting the shear rate or stress sweep. Here we present experimental evidence that paints a contrasting picture. By following another protocol, we show that the substance is required to build up the contact network that culminates in shear thickening. Our study indicates that the contact network results from a cooperative arrangement of smaller clusters of particles. We show that over the period of shear, the suspension goes through continuous shear thickening (CST), then DST, followed by jamming, and finally frictional plastic deformation.

1Support from the Science and Engineering Research Board is gratefully acknowledged
8:24AM Q04.00004 Shear-thickening of a non-colloidal suspension with a viscoelastic matrix , MARCO ELLERO, Basque Center for Applied Mathematics, ADOLFO VAZQUEZ-QUESADA, Universidad Autonoma de Madrid, PEP ESPAOL, Universidad Nacional de Educacion a Distancia (UNED) Madrid, ROGER TANNER, University of Sydney — In this work we study the rheology of a non-colloidal suspension of rigid spherical particles interacting with a viscoelastic matrix. Three-dimensional numerical simulations under shear flow are performed using the smoothed particle hydrodynamics method and compared with experimental data using different elastic Boger fluids. The rheological properties of the Boger matrices are matched in simulation under viscometric flow conditions. Suspension rheology under dilute to semi-concentrated conditions is explored. It is found that at small Deborah numbers, relative suspension viscosities $\eta_r$ exhibit a plateau at every concentration. By increasing $\text{De}$ shear-thickening is observed. Under dilute conditions ($\varphi = 0.05$) numerical results for $\eta_r$ agree quantitatively with experimental data. By increasing the solid volume fraction towards $\varphi = 0.3$, despite the fact that the trend is well captured, the agreement remains qualitative. With regard to the specific mechanism of elastic thickening, the microstructural analysis shows that it correlates well with the averaged viscoelastic dissipation function, requiring a scaling as $\text{De}^{\alpha}$ with an exponent $\alpha$ greater than 2 to take place. Locally, flow regions responsible of the elastic thickening are well correlated to areas with significant extensional component.

8:37AM Q04.00005 A new dimensionless number governing dethickening in orthogonally perturbed shear thickened suspensions , MEERA RAMASWAMY, Department of Physics, Cornell University, ABHISHEK SHETTY, Rheology department, Anton Paar USA, ITAI COHEN, Department of Physics, Cornell University — When concentrated colloidal suspensions are under stress, their viscosity can increase by over an order of magnitude. Previous work has shown that this shear thickened viscosity can be tuned by applying fast oscillatory perturbations orthogonal to the primary shear flows in the system. In this talk, I show that dethickening in the regime where the primary shear flow has fully thickened the suspension, is governed by a single dimensionless number -- the ratio of the orthogonal shear rate amplitude to that of the primary shear rate. In contrast, a second parameter is required to describe the data in the primary shear flow regime where the suspension is thickening. Understanding these parameters will enable better strategies to tune the properties of shear thickening suspensions for applications ranging from 3D printing to the processing of cement.

8:50AM Q04.00006 ABSTRACT WITHDRAWN —

9:03AM Q04.00007 Dynamical behavior of electrореological suspensions , SUCHANDRA DAS, New Jersey Institute of Technology, SRIRAM PILLAPAKKAM, Temple University, NAGA MUSUNURI, Indiana Tech, ISLAM BE-NOUJAGUEF, New Jersey Institute of Technology, EDISON AMAH, Intel Corporation, IAN FISCHER, PUSHPENDRA SINGH, New Jersey Institute of Technology — Electrorheological suspensions are formed by suspending dielectric solid particles in a dielectric liquid. The size of the particles varies between nano and micro meters, depending on the requirements of the intended application. When an electric field is applied, the particles become polarized and form chains and columns which align in the direction of the field, and this increases the viscosity of the electrorheological suspension. This change in the suspension microstructure and viscosity happens within a few milliseconds after the electric field is applied, and when the electric field is removed the viscosity goes back to the original value, which makes the suspension suitable for the applications where a quick response time is desired. The aim of this work is to employ the experimental and direct numerical simulation techniques to study the electrorheological response and the role of parameters such as the particle size and polarizabilities in the process.

9:16AM Q04.00008 Oscillatory shear response of the rigid-rod model in nematic regime¹, GIOVANNANTONIO NATALE, University of Calgary, MARCO DE CORATO, Universitat de Barcelona — Nematic phase of rigid-rod molecules presents rheological complexities given the intrinsic anisotropy of the molecules and spatial variation of an average molecular orientation (director) in the bulk. This microscopic model has been investigated in simple shear flow showing complex dynamics (log-rolling, wagging and tumbling regimes). Oscillatory shear flow is a model transient flow field which introduces a transient and periodic perturbation to the system. Recently, large amplitude oscillatory shear (LAOS) has attracted interest given the rich rheological response that is obtained. However, the interpretation at the microstructural level of the LAOS response is still limited to specific systems. Here we perform numerical simulations of the Doi-Hess equation in oscillatory shear for the molecular orientational distribution function using Brownian dynamics and an expansion in spherical harmonics. A new methodology to switch between the nematic and isotropic orientational state thanks to the transient nature of the flow is proposed. Moreover, oscillatory shear flow is found to be more efficient than the simple shear flow to capture the full microstructural dynamics. This methodology can provide a new strategy for experimental characterization of nematic colloidal suspensions.

¹Dr. Natale acknowledges the support of the Natural Sciences and Engineering Research Council of Canada (NSERC)

9:29AM Q04.00009 Extensional rheology of a dilute suspension of spheres in a dilute polymer solution¹, ARJUN SHARMA, Graduate Student, Sibley School of Mechanical and Aerospace Engineering, Cornell University, Ithaca, NY, 14853, USA, DONALD KOCH, Professor, Robert Frederick Smith School of Chemical and Biomolecular Engineering, Cornell University, Ithaca, NY 14853, USA — We investigate the steady-state rheology of a dilute suspension of spherical particles in a dilute polymer solution modeled by the FENE-P constitutive relation. This work uses a semi-analytical method based on ensemble averaged equations, a perturbation for small polymer concentration and the generalized reciprocal to determine the polymers’ influence on the particle stresslet and the particles’ influence on the polymer stress. In the undisturbed flow, polymers undergo a rapid transition from a coiled to almost fully stretched state at a critical Deborah number (ratio of polymer relaxation to flow time scale) of $D_e = 0.5$. The particle-polymer contribution enhances the averaged stress below $D_{ec}$ due to a strong local stretch enhancement in specific regions of the flow. Above $D_{ec}$, polymers passing around the particle tend to collapse in response to a time history of strain rates which on average are smaller near the particle. These insights are elucidated through a variant of the Finite-time Lyapunov exponent for the Stokes velocity around the sphere. Similar to the undisturbed stress, below $D_{ec}$ we find the particle- polymer contribution to the average stress to be independent of the maximum stretching length (L) and for higher De to scale as $L^2$ for $L \gg 1$.

¹NSF grant 1803156

Tuesday, November 26, 2019 7:45AM – 9:42AM –
Session Q05 Compressive Flows: Supersonic and Hypersonic Flows 204 - Thomas Corke, University of Notre Dame
7:45AM Q05.00001 Reducing shock related unsteadiness of supersonic flow over spiked bodies, JACOB COHEN, DEVARBATA SAHOO, S. K. KARTHICK, Aerospace Engineering, Technion-Israel Institute of Technology, Haifa - 320003, SUDIP DAS, Space Engineering & Rocketry, Birla Institute of Technology, Mesra - 835215 — Spikes, mounted at the stagnation point of a blunt body moving at supersonic speed, are utilized to reduce the forebody drag at the expense of increase in flow unsteadiness. We examine three configurations: flat, hemispherical and elliptical blunt bodies. Our goal is to distinguish between the unsteadiness mechanisms associated with each configuration, using experiments and CFD conducted at \( M = 2 \). Drag is measured by in-house built balance, whereas unsteady pressure transducers and high-rate Shadowgraph snapshots, subjected to POD and DMD analysis, are utilized to quantify the flow unsteadiness. The flat blunt case has been investigated in the past and two modes of unsteadiness have been reported: a mild flapping oscillation and a violent axial pulsation. The latter mode is used to validate our results. When a spike is attached to hemisphere/ellipse body, the upstream strong detached bow shock wave is replaced by a system of weaker oblique shocks. The flow then separates over the spike body forming of a separated flow region bounded by an axisymmetric shear layer. Within the shear layer KH vortices are formed and shocklets are attached to these vortices and all move together downstream. We find the separation angle, separation shock and the separated volume to govern the flow unsteadiness. These are further demonstrated by using the hemispherical forebody with hemispherical spike tip.

7:58AM Q05.00002 Instantaneous velocity fields in a hypersonic wake, YIBIN ZHANG, DANIEL RICHARDSON, STEVEN BERESH, KATYA CASPER, MELISSA SOEHNEL, RUSSELL SPILLERS, Sandia National Laboratories — The cold wake behind a slender object in hypersonic flow offers a challenging environment for off-body measurements. Femtosecond Laser Electronic Excitation Tagging (FLEET) is a simple (single-laser, single imaging system) diagnostic that permits flowfield visualization without seeding or physical probes. FLEET optical and imaging parameters are tailored for measurements in the wake of a sharp cone in Mach 8 nitrogen flow, over freestream Reynolds numbers from \( 4 \times 10^6 \) to \( 14 \times 10^6 \). Fluorescing FLEET lines for 2D velocity measurements and crosses for 2D measurements are written into the flow. The signal-to-noise ratio and tagging efficiency are maximized by using third harmonic FLEET at 267 nm, which has never before been implemented in a hypersonic test case. Flow tagging captures prominent wake features such as the separation shear layer, wake turbulence and two-dimensional velocity components.

8:11AM Q05.00003 Stability and dynamics of supersonic axisymmetric bluff-body wake flows, SIMON GALIVEL, SAMIR BENEDDINE, DENIS SIPP, ONERA, SEBASTIEN ESQUIEU, CEA — The dynamics of separated wake flows behind an axisymmetric bluff body, where complex aerodynamic phenomena occur, is of primary importance for a correct estimation of its drag and trajectory. Those phenomena have been widely studied in incompressible flows, showing the existence of two bifurcations in the laminar regime which first steadily breaks the axisymmetry of the baseflow and then generates vortex shedding. The corresponding laminar large scale structures persist in turbulent regime. Unfortunately, an equivalent study for supersonic conditions is absent from the literature. Thus this work focuses on a supersonic wake of such bluff body \( (M = 4) \) in the laminar regime. A detailed study of unsteady simulations shows several instabilities: the breaking of the axisymmetry, complex flapping and rotation of the backflow region and an oscillating expanded region. Global stability analyses are then carried out to identify and study the flow mechanism of these instabilities. It highlights the driving role of the separation shear layer, of the backflow region, and of the expansion fan, and shows that the dynamics is significantly different from its incompressible counterpart.

8:24AM Q05.00004 DNS of transitional hypersonic boundary layers at high enthalpies, MARIO DI RENZO, PARVIZ MOIN, JAVIER URZAY, Center for Turbulence Research, Stanford University — Aerospace vehicles flying at hypersonic speeds are subject to boundary-layer transition, which causes a strong localized increase in wall heat transfer and friction. The influences of air dissociation at high-Mach numbers on the full process, including the non-linear early stages of turbulence, remain mostly unknown, and cannot be easily accessed by linear stability analyses or parabolized stability equations. In this presentation, DNS results of a hypersonic transitional boundary layer of dissociating air at high-enthalpy conditions are discussed, with particular focus on thermochemical effects on peak values of heat and shear stress. These simulations employ a novel task-based high-order solver written in the programming language Regent that is designed for exploiting GPU-based supercomputers.

8:37AM Q05.00005 Frequency-Wavenumber Spectrum of Surface Pressure Fluctuations Induced by High-Speed Turbulent Boundary Layers, JUNJI HUANG, YUCHEN LIU, LIAN DUAN, The Ohio State University — Spatio-temporal structure of the fluctuating pressure field induced from high-speed turbulent boundary layers is analyzed by using a database of direct numerical simulations (DNS). Specifically, DNS are used to examine and compare the frequency-wavenumber spectrum of wall pressure generated by Mach 8 turbulent boundary layers developing spatially over several canonical geometries including a 2-D flat wall, the inner wall of an axisymmetric nozzle, and a sharp slender circular cone. The study provides insights into the scaling of pressure disturbance spectrum with respect to the boundary-layer parameters and the flow configuration. Such information is important to developing physics-based models that could adequately predict the magnitude, frequency content, location, and spatial extent of boundary-layer-induced pressure fluctuations at high speeds for the structural design of high-speed vehicles.
8:50AM Q05.00006 Particle Relaxation Time for Titanium Dioxide in Hypersonic Flow¹, JOSE RODRIGUEZ, University of Illinois at Urbana-Champaign, BRIAN RICE, CHRISTOPHER MCKENNA, Air Force Research Laboratory — A critical component of the experimental technique Particle Image Velocimetry (PIV) is the response time of the seeding particles. Shock waves are commonly found in high-speed flow regimes, resulting in velocity discontinuities between pre- and post-shock regions. Seeding particles experience a time delay in the normal component of velocity as they transit the shock wave, known as particle relaxation time (PRT). Ragni et al. (J. Exp. Fluids, 2011). This delay is a function of the diameter and density of the particle and the surrounding fluid's viscosity. These properties influence the particles' light-scattering ability and Stokes number (a ratio between the particle and fluid time scales) Melling et al. (J. Meas. Sci. Tech, 1997). These properties contribute to imaging quality and how accurately the particles trace the flow. The PRT for TiO₂ across an oblique shock was characterized via 2D PIV conducted in the von Karman Gas Dynamics Facility Tunnel D at Arnold Air Force Base. A 6.5 degree wedge was used to generate the oblique shock wave in a Mach 5 freestream. The PIV data was fit to exponential Stokes drag decay to obtain the PRT. These findings will be compared with results for other particles that are currently being used throughout the community.

¹Air Force Research Laboratory

9:03AM Q05.00007 Dependence of penetration and entrainment on injectant properties for a jet in supersonic crossflow¹, DAN FRIES, DEVESH RANJAN, SURESH MENON, Georgia Institute of Technology — Four gases with different molecular weights and specific heat ratios are injected as circular, sonic jets into a supersonic crossflow (Mach 1.72). The jet fluid concentration distribution is quantified using a solid particle Mie-scattering technique. To account for the freestream Mach number, the bow shock in front of the jet and boundary layer thickness, an effective momentum flux ratio and a penetration relation based on momentum balance between jet and crossflow are used to improve the collapse of jet trajectories. The trajectory results also suggest a systematic influence of injectant properties on penetration that goes beyond what has been considered in the past. Trends are presented and analyzed further by association with Particle Image Velocimetry velocity data. The development of the velocity field especially in the windward shear layer of the jet elucidates changing compressibility effects on jet spreading and crossflow fluid mass entrainment.

¹In part funded by AFOSR

9:16AM Q05.00008 Stationary-Traveling Cross-Flow Transition Study on a Sharp Cone at Mach 6¹, ERIC MATLIS, ALEXANDER ARNDT, THOMAS CORKE, University of Notre Dame, MICHAEL SEMPÈR, US Air Force Academy — Experiments at Mach 6 in the 3-D boundary layer on a right-circular cone at an angle of attack have revealed evidence of a nonlinear (quadratic) interaction between stationary and traveling cross-flow modes that affect the boundary layer transition Reynolds number. The wavenumber of the stationary modes was controlled by passive patterned roughness located at Branch I. A glow-discharge electrode surface actuator was sputter-deposited just upstream of the patterned roughness to excite the traveling modes at a particular frequency. The observed stationary-traveling interaction was documented with miniature Kulite probes and appeared as an azimuthal variation in the amplitude of the traveling cross-flow mode with azimuthal wavenumbers that corresponded to the sum and difference of the azimuthal wavenumbers of the primary stationary and traveling modes. Cross-bicoherence verified a triple phase locking between the two primary modes and the summed mode for both the “critical” and “subcritical” roughness cases.

¹Supported by AFOSR Grant FA9550-15-1-0278

9:29AM Q05.00009 Sensitivity of hypersonic boundary layers to n-periodic surface roughness-element arrays and finite-rate chemistry¹, ATHANASIOS MARGARITIS, Imperial College London, TARANEH SAYADI, Sorbonne University, OLAF MARXEN, University of Surrey, PETER SCHMID, Imperial College London — Finite-rate thermochemical effects have an order-one influence on the macroscopic behavior of hypersonic boundary layers, hence they have to be accounted for in numerical simulations. Flow stability and heat loads are significantly affected by the modelling approaches in such simulations. Highly-parametrized thermochemical models are commonly used, introducing large amount of uncertainty. Furthermore, the effect of surface roughness and its potential interaction with finite-rate chemistry effects remains unexplored. A mathematical framework for linearized analysis of n-periodic systems of roughness elements is developed, allowing us to extract information about wake synchronization from reduced-cost simulations of a single unit or a triplet of units; this modifies the restrictive assumption of single-unit periodicity. An efficient adjoint-based sensitivity analysis is used to identify critical roughness or chemical model parameters, with respect to their effect on output flow quantities. This framework is applied to generic flat-plate boundary layer configurations for validation; extensions to more complex flow configurations are readily feasible. Preliminary results of an n-periodic system analysis for reacting and non-reacting flows will be presented.

¹AFOSR/EOARD FA9550-18-1-0127

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q06 Acoustics: General 205 - Likun Zhang, University of Mississippi

7:45AM Q06.00001 Acoustic radiation force and scattering series expansions for spheres at low frequencies,¹, PHILIP L. MARSTON, Washington State University — Even in the case considered here where spheres are taken to be in inviscid fluids and all mechanisms for power absorption are neglected, it can be helpful to consider leading-order corrections to Rayleigh scattering. Using series expansions of scattering partial-wave phase shifts (that depend on material properties) quantities of interest can be expanded in powers of kR where k is the acoustic wave number and R is the sphere radius. This has been accomplished in a unified way for several cases, though it is necessary for kR to be below all resonances in each case. Situations considered include fluid and solid spheres and empty elastic shells in traveling and standing acoustic plane waves [1, 2]. The method is easily generalized to spheres in certain acoustic beams. There has also been renewed interest in low kR expansions of the quadrupole projection of the acoustic radiation stress based on Rayleigh scattering since that is relevant to the equilibrium shape of nearly-spherical acoustically-trapped objects [3]. [1] P. L. Marston, J. Acoust. Soc. Am. 145, EL39–EL44 (2019). [2] P. L. Marston, J. Acoust. Soc. Am. 146 (accepted). [3] P. L. Marston et al., J. Acoust. Soc. Am. 69, 1499-1501 (1981).

¹Supported by ONR.
Numerical investigation of heat transfer induced by an oscillatory flow within a thermoacoustic engine core\textsuperscript{1}, KAZUTO KUZUKI, SHINYA HASEGAWA, Tokai University — Quantification of heat transfer phenomena in a thermoacoustic engine is a key to the successful improvements in the engine performance. From this point of view, Piccolo and Pistone (Int. J. Heat Mass Tran. 49 (2006), pp. 1631-1642) presented a computational method for heat transfer analysis of the thermoacoustic engine. In their method, a time-averaged energy conservation equation is solved for the acoustic field fixed at thermoacoustic engine core, and for simplification, the acoustic field was assumed to be a standing wave. However, the actual thermoacoustic device is often designed so that a travelling wave could be achieved at the engine core part, and furthermore, there are interactions between gas motion and temperature field. The present study proposes a computational model which can consider such interactions and travelling wave. In this model, both the Rott’s acoustic approximation equation (Thermoacoustics, ASA Press, pp.102) and the time-averaged energy conservation law are simultaneously solved in two-dimensional space. From the results, heat transfer within a thermoacoustic engine core is discussed.

\textsuperscript{1}Financial support by JST/ALCA (Advanced Low Carbon Technology Research and Development Program: Grant number JPMJAL1305)

Linear and nonlinear propagation of single-frequency dissipative waves in ducts with slowly-varying cross section\textsuperscript{1}, PABLO L. RENDON, ICAT, Universidad Nacional Autónoma de Mexico, NIGEL PEAKE, DAMTP, University of Cambridge — Finite-amplitude sound waves propagating in ducts are subjected to a variety of competing effects. In the context of the acoustics of wind instruments, nonlinear steepening is known to occur in trombones and trumpets when these instruments are played loudly, and in aircraft engine intakes buzz-saw noise is also associated with nonlinear propagation. The main dissipative mechanisms in these ducts tend to be losses at the duct walls rather than viscous thermal attenuation. These losses are due to a Stokes boundary layer which oscillates as the waves pass. In both these contexts, the duct cross-section varies slowly, and this typically means that the wavenumber will change along the length of the duct. We use a WKBJ method to obtain equations which describe acoustic wave propagation subject to combinations of these effects, and for ducts of different geometries.

\textsuperscript{1}The authors acknowledge support from DGAPA-UNAM PAPIIT IG100717, PASPA-UNAM, and CONACYT.

3D Thermoacoustics in a Microwave Plasma\textsuperscript{1}, SETH PREE, JOHN P KOU LAKIS, SETH PUTTERMAN, University of California Los Angeles — Pulsed microwaves directed at an acoustic cavity filled with partially ionized gas may generate intense sound fields. These fields have been shown to center and confine the hottest portions of the gas to the center of a spherical cavity via a generalization of acoustic radiation pressure we have called the pycnologic acoustic force. Because partially ionized gas is luminous, the sound field is rendered visible by luminosity and temperature oscillations caused by the acoustic field’s periodic adiabatic compression. This observation indicates that the microwave absorptivity of the gas may also fluctuate as sound passes through it. If microwave absorption increases in phase with the acoustic compression, the conditions for acoustic amplification may be met. This would allow the generation of high amplitude sound and possibly confinement with a continuous wave microwave source. We will describe the apparatus, present evidence of acoustic plasma confinement, and outline the theoretical conditions for microwave plasma thermoacoustics. We will also discuss how sustaining such a high amplitude sound field and the pycnologic acoustic force with this 3D thermoacoustic effect may enable a new laboratory model of convection in a central force.

\textsuperscript{1}This research was developed with funding from the Defense Advanced Research Projects Agency (DARPA) under award D19AP00015.

Experimental study of one-dimensional acoustic metamaterials of sound-soft inclusions\textsuperscript{1}, CAMILA HORVATH, MARIA LUISA CORDERO, DFI, FCFM, Universidad de Chile, AGNÉS MAUREL, Institut Langevin, ESPCI ParisTech, CNRS UMR 7587. — We are studying the properties of a one-dimensional acoustic metamaterial. The metamaterial consists of a periodic array of rectangular cross section pillars, made of a sound penetrable material (air) built-in a medium with higher acoustic impedance (Polydimethylsiloxane). We are focused on the acoustic response, when the periodicity of the array is several orders of magnitude lower than the wavelength of the incident wave. The problem is being studied experimentally and theoretically, using homogenization methods (Marigo, 2018, RSPA; Maurel, 2014, JASA) to simulate acoustic response of the array and comparing these results with the experimental measurements. Using lithography and soft lithography techniques, we fabricate the metamaterial, shaping a microscale periodical structure of rectangular cross section air bubbles within a Polydimethylsiloxane media. In order to reproduce a one-dimensional array, the air bubbles are pillars with a length, normal to its cross section, four orders of magnitude larger than the periodicity of the structure. We study experimentally the acoustic wave transmitted and reflected by the metamaterial. We find that it behaves as a perfect reflector for a wide range of frequencies.

\textsuperscript{1}Ph.D. student finance by CONICYT National PhD-fellowship (2016) No. 21161479. Funding from the millennium nucleus physics of active mater, FONDEQUIP EIQM140055 and Fondecyt grant No. 1170411 are acknowledge.

Analysing the structure of the acoustic analogy based-high frequency Greens function in non-axi-symmetric sheared flows via a Ray tracing solver\textsuperscript{1}, SARAH STIRRAT, MOHAMED APSAR, University of Strathclyde, ADRIAN SESCU, Mississippi State University — The chevron nozzle remains a popular approach aimed at reducing jet noise. It works by breaking up large turbulence structures and by increasing mixing, but it also effects propagation of sound. In this study, we investigate the effect of chevron-type mean flow in an acoustic analogy model where the wave propagation reduces to the solution of the Rayleigh equation and is calculated using a ray theory model for a jet represented by a transversely sheared mean flow. Since the generalised acoustic analogy (GAA) shows that the acoustic pressure is given by the convolution product of a rank-2 tensor propagator and the fluctuating Reynolds Stress, we determine the propagator (that is related to the vector adjoint Green’s function of the linearised Euler operator) using the high frequency Ray theory developed by Goldstein (J. S. V., Vol. 80, p. 499, 1982) under an isotropic model of the fluctuating Reynolds stress. We calculate the Rayleigh equation Greens function at high frequencies for a series of chevron mean flow patterns with multiple lobes. Our results reveal that the chevron jet introduces much more non-periodic spatial modulation of the Greens function with a local minima in amplitude within the jet. We conclude by discussing how this explains the observed reduction in sound.

\textsuperscript{1}We thank Dr. S.J. Leib (Ohio Aerospace Institute) for use of his ray-tracing code.
9:03AM Q06.00007 Outer streaming within a two-dimensional channel

KYLE PIETRZYK, ILENIA BATTIATO, Stanford University — Acoustic streaming is the net time-averaged flow that results from nonlinearities in an oscillating flow. Extensive research has sought to identify different physical mechanisms and regimes of acoustic streaming in systems of various geometries. While streaming in a channel maintains a simple geometry, it requires an appropriate set of scales to capture the multiple regimes of streaming that can occur. This study aims to define a set of scales and dimensionless numbers for general outer acoustic streaming within a channel. The chosen scales are validated through the recovery of slow streaming equations describing Rayleigh streaming in a channel and Eckart streaming for the case of infinitely far away channel walls. Using the scales and the time-averaged momentum equations, fast streaming is then analyzed and nonlinear Reynolds numbers, which indicate whether the streaming is nonlinear or linear, are found. With this analysis, a brief procedure for identifying regimes of acoustic streaming within a channel is provided for future analyses involving streaming in complex multi-scale systems.

1Stanford Graduate Fellowship in Science and Engineering

9:16AM Q06.00008 Acoustic propagation in random media using polynomial chaos expansions

ALEXANDRE GOUPY, CMLA, ENS Paris-Saclay, France, CHRISTOPHE MILLET, CEA & CMLA, ENS Paris-Saclay, France, DIDIER LUCOR, LIMSI, France — Sound propagation in the atmosphere is highly dependent on the information to specify the waveguide parameters. For real-world applications, there is considerable uncertainty regarding this information, and it is more realistic to consider the wind and temperature profiles as random functions, with associated probability distribution functions. Even though the numerical methods currently-in-use allow accurate results for a given atmosphere, high dimensionality of the random functions severely limits the ability to compute the random process representing the acoustic field, and some form of sampling reduction is necessary. In this work we use polynomial chaos (gPC)-based metamodels to represent the effect of large-scale features onto the acoustic normal modes. The impact of small-scale atmospheric structures is modelled using a perturbative approach of the coupling matrix. This two-level approach allows to estimate the statistical influence of each mode as the frequency varies. An excellent agreement is obtained with the gPC-based propagation model, with a few realizations of the random process, when compared with the Monte Carlo approach, with its thousands of realizations.

9:29AM Q06.00009 Reduced Order Modeling of Spray Flame Response to Harmonic Velocity Fluctuations

VISHAL ACHARYA, Georgia Institute of Technology — Modern combustion systems are all susceptible to thermoacoustic combustion instabilities. To understand these instabilities, reduced order models are developed for the dynamics of the flame when subjected to various source fluctuations, such as due to velocity, mixture ratio, pressure etc. Prior research has significantly focused on modeling for gaseous premixed flame dynamics with recent research also increasing focus on gaseous non-premixed diffusion flames. However, reduced order modeling for spray flames has received no attention and thus this work presents a modeling framework for the dynamics of spray flames with a focus on the velocity coupled response. The response is characterized using the Flame Transfer Function (FTF). The paper uses the classical Burke-Schumann diffusion flame configuration as a basis with the fuel introduced in the form of a spray of liquid droplets. The space-time dynamics in the model uses the fast-chemistry limit applied to the mixture fraction equation for both the gaseous and liquid phases. These equations are coupled through evaporation of the liquid droplets and results in new control parameters such as a vaporization Damkohler number in addition to parameters pertaining to the spray and droplets themselves. Collectively, the effects of these new parameters on spray flame dynamics can be understood.

Tuesday, November 26, 2019 7:45AM - 9:42AM  
Session Q07 Non-Linear Dynamics and Chaos III 211 - Harry Swinney, University of Texas

7:45AM Q07.00001 Transport by vortices formed by breaking internal waves on a continental slope

HARRY SWINNEY, University of Texas at Austin, GUILHERME SALVADOR-VIEIRA, MICHAEL ALLSHOUSE, Northeastern University — Oceanic internal waves generated by tidal flow over bottom topography can transport energy for thousands of kilometers, but in the open ocean material transport (a second-order effect) is not significant. However, when an internal wave impinges on a continental slope, it forms coherent vortices (called boluses) that can trap and transport particles and biota along the slope. The magnitude of such transport in the oceans is not known. While most previous studies examined bolus transport for model systems consisting of two layers of uniform density, the present laboratory experiments and numerical simulations examine how transport by boluses depends on the thickness of the pycnocline, the region in which the fluid density changes rapidly with depth. We find that bolus size, upshele displacement, and maximum available potential energy produced are optimized for a particular pycnocline thickness and are significantly larger in continuously stratified fluids than in two-layer models. Linking the observed transport relationships to ongoing observations of coastal boluses should provide more accurate estimates of the importance of bolus transport for global coastal ecosystems.

7:58AM Q07.00002 Bacterial diodes: Rectified transport of swimming cells in porous media flow

JEFFREY GUASTO, NICOLAS WAISBORD, Tufts University — Directed motility enables swimming microbes to navigate their porous habitats for resources, where self-propulsion competes with fluid flow to affect processes ranging from disease transmission and bioremediation. Despite this broad importance, how directed motility affects the self-transport and dispersion of microswimmers in flow through constricted porous remains unknown. Focusing on magnetotactic bacteria in a microfluidic porous medium, we show that upstream oriented cells, directed by a magnetic field, are localized and trapped in vortical orbits at a constriction. Vortical cell localization results in three distinct regimes of rectified bacterial conductivity through a throat, akin to a ‘bacterial diode’, whereby cells swim upstream, become trapped within a pore, or are advected downstream with increasing flow speed. Langevin simulations reveal that the trapping regime results in near-complete transport suppression, while ephemoral trapping in the downstream regime enhances dispersion. We also show that vortical cell localization persists in three-dimensional flow through a packed microfluidic bed, emphasizing the relevance of this phenomenon in realistic hydraulic networks.

8:11AM Q07.00003 Several uses of Hopf bifurcations in devices, biology and fluids

RANDALL TAGG, University of Colorado Denver — A casual conversation (c1987) with Jerry Gollub about the marginal oscillator detector used in early magnetic resonance experiments launched a fruitful direction of inquiry about Hopf bifurcations by several undergraduates and some graduate students. This includes use of a Wien bridge oscillator as a sensitive detector, uncovering some new features of this well-known circuit including aspects of dynamics on a torus near the initial bifurcation and explorations of noise near the bifurcation. Mention will be made of other perhaps unexpected aspects and potential uses of Hopf bifurcations (or closely related phenomena) in biology and fluids.
about the granular flow characteristics play a central role in the variability of the sintered microstructure.

using the granular feedstock model to initialize a phase-field SLS model. Sensitivity analysis of pore structure features shows the assumptions

These three factors are shown to have significant effects on sub-optimal packing fractions of feedstock material, which suggest assumptions of

Contributions are presented which take into account particle characterization, the mechanical deposition process, and inter-particle cohesive forces.

irregular particles which are then fused by exposure to a laser. Physical models to predict microstructure variability, especially pore characteristics,

ratory, CHEMICAL DYNAMICS INITIATIVE TEAM — Selective laser sintering (SLS) additive manufacturing processes distribute layers of

stocks distributions

forces and torques in the system undergoing cyclic forcing.

irreversible. The translational and rotational displacements are only weakly correlated, indicating that rotational motion depends on more subtle

tracking displacements but also able to follow rotations. We observe that for moderate compression amplitudes, up to one bead diameter, the

under cyclic compression, Using transparent acrylic beads with cylindrical holes and index matching techniques, we are not only capable of

granular flows. In the work presented here we analyze the reversibility of both translation and rotations of granular materials in three-dimensions

8:37AM Q07.00005 Inverted Peristaltic Pumping in Brain?1, BRUCE GLUCKMAN, FRANCESCO COSTANZO, Penn State University — Clearance of brain waste products — metabolites and macromolecules — has been implicated in diseases such as Alzheimer’s Disease. Brain does not have a lymphatic system, so the mechanics of clearance are unknown. The glymphatics model — that traveling diameter variations of penetrating cerebral arterioles due to heart-driven blood pressure pulsations induces an inverted peristaltic pumping of fluid into and through brain — is an attractive model. Here we investigate through rigorous Arbitrary Lagrangian-Eulerian fluid structure interaction modeling different physical formulations and relevant parameters, and whether the biological constraints of the system could support significant flows. Special attention is given to the explicit boundary conditions and values of the problem. The flowrates generated can be significant under many conditions, but not within biophysically relevant pressure differences that otherwise deform the brain tissue.

1Retired from full-time employment most recently at Five Colleges, Incorporated in Massachusetts

8:50AM Q07.00006 Stickslips with Granular Particles, JC TSAI, Inst. of Physics, Academia Sinica, CHENG-EN TSAI, National Central Univ. — We study the emergence of stick-slip instabilities with densely packed granular particles that are sheared continuously. Such behavior occurs in a route of transition between suspension dynamics and plastic flows, as a result of changing the shear rate. Counterpart experiments are designed in order to identify the origin of stick-slips between two particles. We discuss the possible connection between the system-sized avalanches and the local instabilities.

9:03AM Q07.00007 Spatially resolved dynamical transitions in shear thickening fluids1, JEFFREY URBACH, VIKRAM RATHEE, DANIEL BLAIR, Department of Physics, Georgetown University — Dense particulate suspensions exhibit a dramatic increase in average viscosity above a critical, material-dependent shear stress, but the microscopic origins of this shear thickening (ST) remains poorly understood. Using boundary stress microscopy (BSM), we have directly measured the spatially resolved surface stresses during ST, and reported clearly defined dynamic localized regions of substantially increased stress that appear intermittently at stresses above a critical value [1]. Here we present measurements that reveal localized transitions to a fully jammed solid-like phase (SLP) that makes direct contact with the shearing boundaries. The SLPs fracture, bifurcate, and interact in a complex manner that depends on the measurement conditions (constant shear rate vs constant stress). These results demonstrate the ability of BSM reveal rich spatiotemporal dynamics of the thickening transition that are not observable in standard bulk rheology.1 V. Rathee, D. L. Blair, J. S. Urbach, PNAS 114, 8740 (2017).

1Supported by NSF DMR-1809890

9:16AM Q07.00008 Exotic patterns in Faraday waves, LAURETTE TUCKERMAN, PMMH-CNRS-ESPCI-Sorbonne, RAHUL AGRAWAL, IIT Bombay, ALI-HIGO EBO-ADOU, Centre d’Etudes et de Recherche de Djibouti, LYES KAHOUADJI, Imperial College London, JUAN MARIN, NICOLAS PERINET, Universidad de Santiago de Chile, JALEL CHERGUI, DAMIR JURIC, LIMSIC-CNRS, SEUNGWON SHIN, Hongik University — For the Faraday instability, by which standing waves appear on the free surface of a vertically vibrated fluid layer, the wavelength is controlled by the forcing frequency rather than by the fluid depth, making it easy to destabilize multiple wavelengths everywhere simultaneously. In the 1990s, this technique was used to produce fascinating new phenomena such as quasipatterns by Edwards & Fauve and superlattices by Gollub, Pier & Kudrolli. This in turn sparked a renaissance of interest in Faraday waves, leading to new mathematical theories and numerical simulations. We will discuss some of the exotic patterns found in recent numerical simulations, such as quasi-hexagons alternating with beaded stripes, a supersquare divided into four subsquares with synchronized diagonal blocks, Platonic solids alternating with their duals while drifting, and a twisted sheared secondary instability of square waves.

9:29AM Q07.00009 Rotational Dynamics of Dense Granular Systems, WOLFGANG LÖSERT, University of Maryland College Park — This study builds on my postdoctoral work in the Gollub lab on particle tracking based analysis of dense granular flows. In the work presented here we analyze the reversibility of both translation and rotations of granular materials in three-dimensions under cyclic compression. Using transparent acrylic beads with cylindrical holes and index matching techniques, we are not only capable of tracking displacements but also able to follow rotations. We observe that for moderate compression amplitudes, up to one bead diameter, the translational displacements of the beads after each cycle become mostly reversible after an initial transient. However, granular rotations are irreversible. The translational and rotational displacements are only weakly correlated, indicating that rotational motion depends on more subtle changes in contact distributions. We are able to observe these changes in particle dynamics simulations. Simulations allow us to assess the forces and torques in the system undergoing cyclic forcing.

Tuesday, November 26, 2019 7:45AM - 9:29AM — Session Q08 Granular Flows II 212 - Richard Lueptow, Northwestern University

7:45AM Q08.00001 Granular flow model for stochastic additive manufacturing feedstock distributions, WILLIAM ROSENTHAL, AMANDA HOWARD, FRANCESCA GROGAN, Pacific Northwest National Laboratory, CHEMICAL DYNAMICS INITIATIVE TEAM — Selective laser sintering (SLS) additive manufacturing processes distribute layers of irregular particles which are then fused by exposure to a laser. Physical models to predict microstructure variability, especially pore characteristics, are restricted by unrealistic assumptions of powder regularity and the deposition process. 2D and 3D models for SLS feedstock particle distributions are presented which take into account particle characterization, the mechanical deposition process, and inter-particle cohesive forces. These three factors are shown to have significant effects on sub-optimal packing fractions of feedstock material, which suggest assumptions of sphericity and optimal packing may lead to inaccurate results if used in sintering models. The impact of these assumptions is illustrated by using the granular feedstock model to initialize a phase-field SLS model. Sensitivity analysis of pore structure features shows the assumptions about the granular flow characteristics play a central role in the variability of the sintered microstructure.
7:58AM Q08.00002 Rayleigh-Taylor and Rayleigh-Benard like instabilities in two-size two-densities granular flow , UMBERTO D’ORTONA, M2P2, CNRS, Aix-Marseille Univ., Centrale Marseille, NATHALIE THOMAS, CNRS, Aix-Marseille Univ — Rayleigh-Taylor and Rayleigh-Benard instabilities are among the most classical hydro-dynamical instabilities. In both cases, the buoyancy is responsible for the instability. When put into motion, dry granular materials behave as liquids. The granular segregation induces a migration of large particles to the surface, while dense particles move to the bottom. Dense and large particles migrate to the top or the bottom depending on size and density ratios. Here, large and dense particles are chosen such that they migrate to the surface. We show experimentally and numerically that a layer of dense particles put above a layer of less dense particles develops a Rayleigh-Taylor instability while flowing. If dense particles are also larger and subject to upward segregation, the two-layer system destabilizes as well. Furthermore, if the system is initially made of one homogeneous layer, the segregation first induces the formation of a dense layer at the surface which destabilizes later. This self-driven Rayleigh-Taylor instability occurs for various density and size ratios. While time evolves, a pattern of parallel stripes forms with convection cells analogous to Rayleigh-Benard cells. The motor of the instability is the granular segregation.

8:11AM Q08.00003 Interplay between hysteresis and nonlocality in granular flows . SAVIZ MOWLAVI, KEN KAMRIN, MIT — The jamming transition in granular materials is well-known to exhibit hysteresis, wherein the level of shear stress required to trigger flow is larger than that below which flow stops. From a rheological standpoint, such behavior is typically modeled as a nonmonotonic flow rule. However, the rheology of granular materials is also nonlocal due to cooperativity at the grain scale, leading to increased strengthening of the flow threshold as system size is reduced. We investigate how these two effects — hysteresis and nonlocality — couple with each other by incorporating nonmonotonicity of the flow rule into the nonlocal granular fluidity (NGF) model, a nonlocal continuum model for granular flows. Comparing predictions of the model with discrete element simulations in the case of planar shear flow with gravity, we show that the inclusion of nonlocal effects is key to explaining certain features of the hysteretic solid-liquid transition as the applied stress is ramped up and down.

8:24AM Q08.00004 Transitional Granular Packing: Rate-dependent Brittleness . CHENG-EN TSAI, National Central Univ. , JC TSAI, Institute of Physics, Academia Sinica — We discover a route of transition over driving rate that bridges two classic regimes of granular dynamics: fluid-lubricated suspension on the fast end, against the largely plastic regime at the slow limit. Here, densely packed centimeter-sized PDMS particles submerged in fluid are sheared at variable but strictly constant rates. Fluctuations on multiple components of boundary force reveal a transitional regime exhibiting brittle failure of the packing at the intermediate driving rates, accompanied by evidence from simultaneous internal imaging. Rate-dependent statistical distribution of avalanches reveals the development of ductility toward the slow limit.

8:37AM Q08.00005 An investigation of mixing ratio effects on a Couette cell granular flow using magnetic particle tracking method1 , XINGTIAN TAO, HUIXUAN WU, UNIVERSITY OF KANSAS — Optical based methods are abundant in flow measurements; however, they can hardly be used in opaque environment. Therefore, a non-optical based particle tracking method — Magnetic Particle Tracking (MPT) — is investigated and three algorithms of data processing are analyzed. These algorithms are sequential quadratic program, extended Kalman filter (EKF), and particle filter. The reconstructed position and orientation of the magnetic tracer is compared with high-speed camera image result, and the accuracy of MPT with EKF algorithm is in the order of 0.6% in position and 1.5 degree in orientation. This technique is applied to study a sheared dense granular mixture in a Couette cell. The mixture comprises of spheres and cylinders (aspect ratio equals to 1). The trajectory and angle alignment of the tracer particle is reconstructed, and its distribution in the Couette cell is depicted. The effect of the mixing ratio on this Couette cell system is characterized by using finite time Lyapunov exponent between neighboring trajectories.

1University of Kansas General Research Fund

8:50AM Q08.00006 Segregation force in granular flows: From single intruders to bidisperse mixtures , RICHARD M. LUEPTOW, YIFEI DUAN, LU JING, JULIO M. OTTINO, PAUL B. UMBANHOWAR, Northwestern University — Recent studies have focused on the size segregation force on a single large intruder particle in granular flows. However, a generalized scaling of the force is still lacking for combined size and density segregation as well as for mixtures (rather than a single intruder). Here we first measure the segregation force on a single intruder in DEM simulations using a spring-based force meter and provide a universal scaling law of the segregation force that predicts whether the intruder will rise or sink depending only on the size and density ratios. Interestingly, the scaled force does not increase monotonically but decreases at large size ratios, explaining experimental observations that very large intruders sink. Then we extend the measurement to bidisperse particle mixtures of varying concentration. The resulting scaling law enables prediction of the segregation direction (rise or sink) of each particle species for varying size ratio, density ratio, and species concentration. Surprisingly, the segregation may invert as the species concentration changes because the segregation force depends on the species concentration. The predictions are validated with DEM simulations and experiments in various flow configurations.

9:03AM Q08.00007 Modeling segregation pattern formation in biaxial spherical tumbler flow , MENGQI YU, PAUL UMBANHOWAR, JULIO OTTINO, RICHARD LUEPTOW, Northwestern University — Flow of size bidisperse granular particle mixtures in a half-full spherical tumbler rotating about two perpendicular axes exhibits segregation patterns that can be observed through the transparent tumbler wall. The patterns resemble predictions based on dynamical systems analysis including non-mixing structures and unstable manifolds, but also depend on the underlying flow field, relative strength of segregation, and collisional diffusion. Discrete element method (DEM) simulations enable precise characterization of the three-dimensional structures of the segregation pattern and statistical analysis of particle movement in the flow. Axial drift of large particles during rotation about a single axis results in migration toward double bands near the rotation poles. At the same time, chaotic advection redistributes large particles in regions outside of non-mixing structures. As a result of both mechanisms, large particles accumulate in regions where axial bands coincide with non-mixing structures. Comparison of particle trajectories in size bidisperse and monodisperse mixtures provides further insight into the interaction between segregation, diffusion, and the underlying flow field that results in pattern formation.
multi-turbine array at an operational wind farm where it statistically significantly increased the power production. The analytic gradient-based steering of wind turbines thus allows for wake steering and can be successfully used for wind farm control and overall power optimization. Floating wind turbines operate in a dynamic state of misalignment because of surge, heave, yaw or tilt motions from wave and wind dynamics. In this work, wake deflection from static tilt is quantified in a scaled wind tunnel experiment to verify the importance of such turbine misalignments in floating wind farms. Cross-plane stereo particle image velocimetry measurements are performed for a scaled wind turbine rotor model, in a sheared boundary layer flow. Wake deflection caused by a counter-rotating vortex pair in the wake of tilted wind turbines is observed. In a sheared boundary layer flow, and with the presence of ground obstruction, a non-symmetrical behavior is observed, for positive and negative tilt, resulting in either a flying or crashing wake.

This work was supported by the National Science Foundation through grant #CBET-1351411 and by National Science Foundation award #1539070, Collaboration Leading Operational UAS Development for Meteorology and Atmospheric Physics (CLOUDMAP).

Fernando Cunez is grateful to FAPESP (grant no. 2016/18189-0) and Erick Franklin to FAPESP (Grants No. 2016/13474-9 and No. 2018/14981-7), to CNPq (Grant No. 400284/2016-2) and to FAPEX/UNICAMP (Grants No. 2210/18 and No. 2112/19) for the financial support provided.

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q09 Wind Turbines: Wakes

7:45AM Q09.00001 Tilt-induced wind turbine wake shape characterization1, JULIAAN BOSSUYT, NASSEM ALI, RYAN SCOTT, RAUL BAYOAN CAL, Portland State University — The wakes of yaw-misaligned wind turbines have been shown to deflect downstream by the formation of a counter-rotating vortex pair, resulting in wake curling. Intentional misalignment of wind turbines thus allows for wake steering and can be successfully used for wind farm control and overall power optimization. Floating wind turbines operate in a dynamic state of misalignment because of surge, heave, yaw or tilt motions from wave and wind dynamics. In this work, wake deflection from static tilt is quantified in a scaled wind tunnel experiment to verify the importance of such turbine misalignments in floating wind farms. Cross-plane stereo particle image velocimetry measurements are performed for a scaled wind turbine rotor model, in a sheared boundary layer flow. Wake deflection caused by a counter-rotating vortex pair in the wake of tilted wind turbines is observed. In a sheared boundary layer flow, and with the presence of ground obstruction, a non-symmetrical behavior is observed, for positive and negative tilt, resulting in either a flying or crashing wake.

1Work supported by the Belgian American Educational Foundation (B.A.E.F.)

7:58AM Q09.00002 Field experiment of wind farm power optimization through wake steering1, MICHAEL HOWLAND, SANJIVA LELE, JOHN DABIRI, Stanford University — Due to greedy individual wind turbine operation, aerodynamic wakes reduce total wind farm power production, thereby increasing the cost of electricity for this resource. Considering the wind farm as a collective, we designed a wake steering control method to increase the power production of wind farms. The method was tested in a multi-turbine array at an operational wind farm where it statistically significantly increased the power production. The analytic gradient-based wind farm power optimization methodology developed can optimize the yaw misalignment angles for large wind farms on the order of seconds.

1MH is funded through the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1656518 and the Stanford Graduate Fellowship.

8:11AM Q09.00003 Development of wake centerline detection algorithms for the study of wind turbine wake meandering1, NICOLAS COUDOU, Université catholique de Louvain, Université de Mons, von Karman Institute for Fluid Dynamics, LAURENT BRICTEUX, Université de Mons, JEROEN VAN BEECK, von Karman Institute for Fluid Dynamics, PHILIPPE CHATELAIN, Université catholique de Louvain — The low-frequency oscillatory motion of wind turbine wakes, also known as wake meandering, is crucial in wind farms as it increases fatigue loads on downstream turbines. The study of this phenomenon requires, as a first step, the determination of the position of the wake. The wake centroid tracking proposed in this work is based on the computation of the wind ipower inside a disk of a diameter equal to the rotor diameter and shifted in a cross-flow plane. The wake center corresponds to the disk position for which the available power is minimum. Mathematically, it consists in locating the maximum of the convolution between the wake centerline can then be obtained by repeating this technique in each cross-flow plane in the wake of a machine. This method being sensitive to strong local minima of wind speed occurring for high level of inflow turbulence, an improved method based on a 3D convolution is proposed in this work. These techniques are applied to the data obtained from Large-Eddy simulations of the NREL 5-MW wind turbine subjected to synthetic turbulent inflows. The computations were performed with a vortex-particle mesh code, the presence of the wind turbine rotor being accounted for through immersed lifting lines.

1Nicolas Coudou is funded by FRIA, Belgium.

8:24AM Q09.00004 Horizontal Axis Turbine Wake Measurements Using Unmanned Aerial Vehicles1, STEWART NELSON, CHRISTOPHER HEINTZ, University of Kentucky, LUKE NORMAN, RUPP CARRIVEAU, University of Windsor, SEAN BAILEY, University of Kentucky — We use up to four highly instrumented, semi-autonomous unmanned aerial vehicles (UAVs) to measure the wakes shed by horizontal axis wind turbines during a one-week measurement campaign conducted at a line of four turbines located in a wind farm in Southern Ontario, Canada. During this campaign we investigated both the evolution of the wakes during a morning boundary layer evolution, and under the effects of wake steering. For the wake steering experiments, one turbine was yawed by 30 degrees with respect to the mean wind. In this talk, we will present results from the UAVs, which acquired horizontal wind velocity profiles across the turbine wakes at hub height, and with 0.3 m horizontal resolution. We also relate these wake measurements with Supervisory Control and Data Acquisition (SCADA) and tower strain data acquired concurrently to the measurements.

1This work was supported by the National Science Foundation through grant #CBET-1351411 and by National Science Foundation award #1539070, Collaboration Leading Operational UAS Development for Meteorology and Atmospheric Physics (CLOUDMAP).
The effects of turbulence on wind turbine performance (thrust and power generation) are often assumed based on analytical considerations, and studies focusing on actual measurements or computations are scarce. For this reason, a measurement campaign was carried out to parametrically study the effect of characteristics such as turbulence intensity and integral scale on the thrust and power generated by a model-scale wind turbine. An active grid has been used to systematically vary the properties of the incoming turbulent flows, generating integral length scales ranging from one order of magnitude smaller to slightly larger than the turbine diameter. Likewise, it has been possible to vary the free-stream turbulence intensity in the range of 0.5% to more than 12%. The model is a speed-controlled wind turbine driven by the incoming flow, moving a permanent magnet DC generator used to actively control the turbine speed and to measure the torque generated by the turbine rotor. The forces on the turbine are measured with a load cell force balance. Rotor geometries with a NACA 63 series aerofoil and different diameters (150, 180 and 200 mm) are tested. Presented data will focus on the variation in power and thrust coefficients with the turbulence properties of the incoming flow.

Kinematic Shear Stress Budget and Relaxation Time-Scales in a Spatially Heterogeneous Canopy Turbulence: an Application to Finite Sized Wind Farms

The wake of a model-scale, horizontal-axis wind turbine was investigated at $3 \times 10^6 < Re_\theta < 8 \times 10^6$ with uniform inflow and with varying degree of free stream turbulence. Data was collected in the High Reynolds Number Test Facility (HRTF) at Princeton University, which enables extremely high Reynolds numbers due to its ability to pressurize the working fluid, air, up to 3500psi. Streamwise velocity measurements were obtained using a nano-scale thermal anemometry probe (NSTAP), allowing for a temporal resolution of 200kHz, and spatial resolution of 60\mu m. Mean and variance profiles were studied at several downstream distances of $0.35 < x/D < 5.1$. These spanwise profiles show the evolution of the turbine's tip vortices as they advect downstream. Freestream turbulence was injected upstream of the turbine using a turbulence generating grid, and the effects of the freestream turbulence on the turbine's wake and performance are investigated.

Effects of Freestream Turbulence on Wind Turbine Wakes

The wake of a model-scale, horizontal-axis wind turbine was investigated at $3 \times 10^6 < Re_\theta < 8 \times 10^6$ with uniform inflow and with varying degree of free stream turbulence. Data was collected in the High Reynolds Number Test Facility (HRTF) at Princeton University, which enables extremely high Reynolds numbers due to its ability to pressurize the working fluid, air, up to 3500psi. Streamwise velocity measurements were obtained using a nano-scale thermal anemometry probe (NSTAP), allowing for a temporal resolution of 200kHz, and spatial resolution of 60\mu m. Mean and variance profiles were studied at several downstream distances of $0.35 < x/D < 5.1$. These spanwise profiles show the evolution of the turbine's tip vortices as they advect downstream. Freestream turbulence was injected upstream of the turbine using a turbulence generating grid, and the effects of the freestream turbulence on the turbine’s wake and performance are investigated.

Effects of Freestream Turbulence on Wind Turbine Wakes

The wake of a model-scale, horizontal-axis wind turbine was investigated at $3 \times 10^6 < Re_\theta < 8 \times 10^6$ with uniform inflow and with varying degree of free stream turbulence. Data was collected in the High Reynolds Number Test Facility (HRTF) at Princeton University, which enables extremely high Reynolds numbers due to its ability to pressurize the working fluid, air, up to 3500psi. Streamwise velocity measurements were obtained using a nano-scale thermal anemometry probe (NSTAP), allowing for a temporal resolution of 200kHz, and spatial resolution of 60\mu m. Mean and variance profiles were studied at several downstream distances of $0.35 < x/D < 5.1$. These spanwise profiles show the evolution of the turbine's tip vortices as they advect downstream. Freestream turbulence was injected upstream of the turbine using a turbulence generating grid, and the effects of the freestream turbulence on the turbine’s wake and performance are investigated.

Experimental study on the effect of turbulence properties on model wind turbine performance

The effects of turbulence on wind turbine performance (thrust and power generation) are often assumed based on analytical considerations, and studies focusing on actual measurements or computations are scarce. For this reason, a measurement campaign was carried out to parametrically study the effect of characteristics such as turbulence intensity and integral scale on the thrust and power generated by a model-scale wind turbine. An active grid has been used to systematically vary the properties of the incoming turbulent flows, generating integral length scales ranging from one order of magnitude smaller to slightly larger than the turbine diameter. Likewise, it has been possible to vary the free-stream turbulence intensity in the range of 0.5% to more than 12%. The model is a speed-controlled wind turbine driven by the incoming flow, moving a permanent magnet DC generator used to actively control the turbine speed and to measure the torque generated by the turbine rotor. The forces on the turbine are measured with a load cell force balance. Rotor geometries with a NACA 63 series aerofoil and different diameters (150, 180 and 200 mm) are tested. Presented data will focus on the variation in power and thrust coefficients with the turbulence properties of the incoming flow.
7:45AM Q10.00001 Synchronization in periodically forced oscillator flows\textsuperscript{1}, BENJAMIN HERRMANN, STEVEN BRUNTON, University of Washington, RICHARD SEMAAN, Technische Universitat Braunschweig — We investigate the synchronization properties of the turbulent wake past a D-shaped bluff body with periodic Coanda blowing. Time series from experimental measurements of the base pressure are used to study the response of the global vortex shedding mode for different actuation frequencies and amplitudes. Multiple regions of synchronization are found, resulting in the so-called Arnold tongues, where the oscillation frequency of the global mode locks-on to a rational multiple of the forcing frequency. We construct a model using a sparse regression and its structure explains these nonlinear couplings as an anharmonic parametric excitation of the global mode. The model is further analyzed using phase reduction theory, predicting the presence of $2:n$ synchronization and revealing the boundaries of the respective Arnold tongues. We postulate that this phenomena is universal to periodically forced oscillator flows, and arises from resonant wave-triads that transfer energy from the $n^{th}$ harmonic of the forcing, i.e., the parent wave, to the conjugate pair of vortex shedding modes, i.e., the daughter waves with half-frequency and higher wavenumber.

\textsuperscript{1}The work has been supported by the PRIME programme of the German Academic Exchange Service (DAAD) with funds from the German Federal Ministry of Education and Research (BMBF).

7:58AM Q10.00002 Dynamics and stability of confined interfacial droplets, STUART THOMSON, MATTHEW DUREY, MILES COUCHMAN, RUBEN ROSALES, JOHN BUSH, Massachusetts Institute of Technology — Millimeter droplets may bounce on the surface of a vibrating liquid bath or “walk” by means of self-propulsion through a resonant interaction with their own wave field. The propulsive wave force exerted by the bath is balanced by the droplet’s inertia and dissipation in the form of drag. In this talk we present the results of a combined experimental and theoretical study in which we consider the dynamics and stability of droplets confined to an annular ring. Single droplets are observed to destabilize to a pulsating walking state and then to a random-walk like instability as the vibrational acceleration of the bath is increased. When multiple droplets are brought into close proximity, they may bind together to form one-dimensional lattices which exhibit several features characteristic of driven dissipative oscillator systems including periodic oscillations and a striking solitary wave-like instability. Our experimental system provides a highly tunable apparatus to study dynamical systems driven far from equilibrium at the macroscale.

8:11AM Q10.00003 Extention of Arnold’s stability theory for planar viscous shear flows, HARRY LEE, University of Michigan, SHIXIAO WANG, University of Auckland — A viscous extension of Arnold’s inviscid theory for planar parallel non-inflow shear flows is developed and a viscous Arnold’s identity is obtained. Special forms of our viscous Arnold identity are revealed that are closely related to the perturbation’s enstrophy identity derived by Synge (1938) (see also Fraternale et al. 2018). Firstly, an alternative derivation of the perturbation’s enstrophy identity for strictly parallel shear flows is acquired based on our viscous Arnold’s identity. The alternative derivation induces a weight function, inspired by which, a novel weighted perturbation’s enstrophy identity is established that extends the previously known enstrophy identity to include general non-parallel streamwise translation-invariant shear flows imposed with relaxed wall boundary conditions. As an application of the enstrophy identity, we quantitatively investigate the mechanism of linear instability/stability within the normal modal framework. The investigation finds that the critical layer is always a primary source of damping in disturbance’s enstrophy and thus it enhances stability. Moreover, a control scheme is proposed that transitions the wall settings from the no-slip condition to the free-slip condition, through which a flow is quickly stabilized.

8:24AM Q10.00004 Nonlinear stability of wall-bounded viscous flows, SHIXIAO WANG, Auckland University, HARRY LEE, University of Michigan — A viscous extension of Arnold’s inviscid theory for planar shear flows is developed and a viscous Arnold’s identity is obtained. The viscous Arnold’s identity is revealed to be closely related to the perturbation’s enstrophy identity (Synge 1938). The mechanism of linear instability/stability of wall-bounded shear flows has been re-examined by the viscous Arnold’s identity and the perturbation’s enstrophy identity. It was found that the role of non-slip wall boundary condition in a planar shear flow is strikingly different from that in the circular Taylor-Couette flow confined between two concentric rotating cylinders. For the former, the perturbation’s enstrophy is generated at the walls by the non-slip induced flow rubbing effect. For the latter, however, the perturbation’s flow circulation is rigidly fixed by the non-slip wall condition and thus it effectively stabilizes the flow globally under axisymmetric disturbances within a sub-domain of the inviscid linear stability regime. A remarkable feature of the global stability for the circular Taylor-Couette flow is its independence of the $Re$ number.

8:37AM Q10.00005 Stability and bifurcation of a freely-rotating discontinuity in a symmetrically driven square cavity, GONGQIANG HE, FTN Associates, Ltd., Baton Rouge, LA, 70808, CHENGQIANG ZHANG, Massachusetts Institute of Technology, KRISHNASWAMY NANDAKUMAR, Louisiana State University, & The Energy Research Institute, Jinan, PRC — Using a direct-forcing immersed boundary method at fully resolved grid resolutions, we study the interaction between the fluid in a symmetrically driven square cavity (i.e., top and bottom lids sliding at identical velocity), and the response of a rectangular block inside. The block is made thin enough to approximate an ideal discontinuity; it is fixed at the cavity center but can freely rotate. We scanned up to moderate Reynolds numbers using different block length $L$, and a phase diagram (Re, L) is created. For a fixed Reynolds number, a short block stabilizes at the vertical orientation ($\theta = 90$ degree). As the block becomes longer, the vertical orientation loses stability and bifurcates into a pair of new stable orientations that are symmetric regarding the vertical direction. The critical lengths for different Reynolds numbers are found, and the reason for the loss of stability is explained by energy argument and analysis of the flow patterns.

8:50AM Q10.00006 Koopman Control of Point Vortex Dynamics using Invariants, KARTIK KRISHNA, ADITYA NAIR, EURIKA KAISER, STEVE BRUNTON, University of Washington, Seattle — We seek to manipulate the behaviour of a planar system of point vortices governed by the Biot-Savart law, which is often used to simulate fluid flows in an inviscid and incompressible setting. Inspired by recent advances in Koopman operator theory, we recast the original Biot-Savart law in terms of well known invariants of vortex dynamics (e.g. the Hamiltonian). This change of variables helps us obtain a linear representation of nonlinear dynamics and reduces the dimensionality for fluid flows, where the number of vortices is very large. We then leverage tools from control theory to manipulate vortex dynamics using virtual cylinders (vorticity generating actuators). In particular, we show that increasing (decreasing) the Hamiltonian enables the clustering (declustering) of multiple vortices. We are currently extending this work to dissipative flows for discrete vortex control.

9:03AM Q10.00007 ABSTRACT WITHDRAWN
Newtonian liquids in stirred vessels; it is regularly encountered in industrial sectors like healthcare, pharmaceuticals and food processing. Stirred tanks are commonly used in industry to process particles with liquids. A viscoelastic liquid was chosen that had matching refractive index with the suspended particles and a novel combination of PLIF techniques was used to measure the concentration measurement within a Hele-Shaw cell Rayleigh-Benard convection as CO$_2$ dissolves into brine. The velocity is measured using PIV under large depth-of-field and gap-wise Poiseuille flow conditions, making reliable quantitative velocity measurement difficult due to the large velocity gradient across the gap. Previously, particle sorting has been purposeful and validated as a means to resolve this problem. The method flushes particles along the cell until all have migrated to their gap-wise equilibrium plane, thus providing an unambiguous velocity magnitude.

9:29AM Q10.00009 A 3-D Poor Man’s Boltzmann Equation, J. M. MCDONOUGH, University of Kentucky (Retired), H. W. YU, Indiana University Purdue University at Indianapolis — A 1-D “synthetic” distribution function for the poor man’s Boltzmann equation (PMBE) has been studied previously; but real applications must be in three space dimensions. In this presentation we outline derivation of the 3-D PMBE and study bifurcations of the corresponding discrete dynamical system (DDS) for this case. In particular, we first provide a brief single-mode Galerkin derivation of the PMBE, and then present time series, power spectra and regime maps to demonstrate its consistency with expected fluid flow behaviors in particular, existence of Ruelle & Takens, Feigenbaum, and Pomeau & Manneville bifurcation sequences, as well as combinations of these. We also suggest how such a DDS can be used to produce very efficient sub-grid scale synthetic distribution function models for turbulence simulations within a lattice-Boltzmann/large-eddy simulation framework.

Tuesday, November 26, 2019 7:45AM - 9:42AM — Session Q11 Experimental Techniques: Quantitative Flow Visualization II 3B - Joe Katz, Johns Hopkins University

7:45AM Q11.00001 Novel Particle Shadow Tracking Velocimetry Technique, CARLOS ECHEVERRIA, DAVID PORTA, CATALINA STERN, ENRIQUE GUZMAN, UNAM, HIDRODINAMICA Y TURBULENCIA TEAM — We present a novel, non-invasive particle tracking velocimetry technique, called Particle Shadow Tracking Velocimetry (PSTV), which is based on the shadows cast by the process particles conveyed within a gaseous phase. Ad hoc digital filters were designed and implemented, in order to detect these solid particles with a high degree of precision. To show this, the methodology relied on the measurement of an Ultra Depth of Field (UDOF) distance required for the correct determination of the detection volume. The PSTV technique produced accurate velocity field values when tested with a validation experiment. We show that the determination of the volume of study is crucial for the correct measurement of the velocity field. To do so, we studied a biphasic solid-gas flow in the compressible regime; the solid phase consisted of a polydisperse granular material with an average size of 0.3mm.

7:58AM Q11.00002 Experimental investigation of the clustering of particles in non-Newtonian liquids in stirred vessels, GIOVANNI MERIDIANO, WEHLIYE HASHI WEHLIYE, LUCA MAZZEI, PANAGIOTA ANGELI, University College London — The blending of solid particles with liquids is a crucial step in many manufacturing processes; it is regularly encountered in industrial sectors like healthcare, pharmaceuticals and food processing. Stirred tanks are commonly used in industry to process solid–liquid mixtures principally because of their flexibility, relative construction simplicity and availability. Recent studies have proposed stirred tanks as a tool to achieve the separation of the solids suspended in Newtonian liquids (Wu et al. 2015). In this study, the investigation of the clustering of particles in non-Newtonian liquids in stirred vessels is presented. A viscoelastic liquid was chosen that had matching refractive index with the suspended particles and a novel combination of PLIF and PIV/PTV techniques was used to measure the velocity fields of the particles and of the fluid simultaneously as well as the evolution of the concentration of the particles in the tank. It was found that the particles accumulated in the cores of the vortices. In addition, the accumulation speed was found to be dependent on the viscoelastic properties of the fluid and on the diameter of the solid particles.

8:11AM Q11.00003 Development of a velocity and concentration measurement method for CO$_2$ dissolution in brine within a Hele-Shaw cell, MENGYE ZHAO, KENNETH KIGER, University of Maryland, ANKUR KISLAYA, JERRY WESTERWEEL, Delft University of Technology — We perform quantitative velocity and concentration measurement within a Hele-Shaw cell Rayleigh-Benard convection as CO$_2$ dissolves into brine. The velocity is measured using PIV under large depth-of-field and gap-wise Poiseuille flow conditions, making reliable quantitative velocity measurement difficult due to the large velocity gradient across the gap. Previously, particle sorting has been purposeful and validated as a means to resolve this problem. The method flushes particles along the cell until all have migrated to their gap-wise equilibrium plane, thus providing an unambiguous velocity magnitude. However, in order to accomplish the sorting, a large portion of the test cell will not be usable for measurement. We purpose a new method to overcome this limitation. We first conduct PIV without sorting particles to obtain only the velocity direction. This provides the value of the in-plane velocity ratio ($u/v$) at every $x,y$ coordinate pair. We then obtain the CO$_2$ concentration map using fluorescent emission of a Ph sensitive dye. Having mapped the concentration, we combine the $u/v$ information and solve the CO$_2$ advection-diffusion equation to get the value of $u$ and $v$. We demonstrate this method by comparing its result with that from the particle sorting method.
A combined structured planar laser-induced fluorescence (S-PLIF) and particle image velocimetry (S-PIV) method for interfacial and near-wall measurements1. VICTOR VOULGAROPOULOS, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — The application of experimental laser-based methods to obtain spatiotemporally resolved information has been emerging as increasingly important in various fields of fluid dynamics, owing to their ability to provide a holistic picture of the phenomena involved and to play a crucial role in benchmarking numerical models. Two-phase systems consisting of fluids with strong variations in their refractive indices in the visible wavelengths, e.g., any gas-liquid system, have, however, been found to compromise traditional laser-based techniques. These optical techniques become susceptible to laser-light reflections and reflections close to the fluid interface, resulting in erroneous interface location and velocity measurements. In this work, we investigate horizontal gas-liquid stratified pipe flows employing a new structured illumination technique. We perform simultaneous structured planar laser-induced fluorescence (S-PLIF) and structured particle image velocimetry (S-PIV) measurements to obtain phase and velocity fields, by minimising total internal reflections effects encountered in traditional PLIF and PIV measurements. The wave characteristics as well as the near-wall and interfacial flow properties are discussed.

This work was funded by the U.S. Department of Energys Solar Energy Technologies Office under the Gen 3 CSP program. Sandia National Laboratories is a multimission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energys Nuclear Security Administration under contract DE-NA0003525.

1Support from TOTAL and the Transient Multiphase Flows Consortium is gratefully acknowledged.

Velocity Vector Field Extraction from High Speed Thermograms through Particle Image Velocimetry Tools1. GUILLERMO ANAYA, JESUS ORTEGA, IRMA VAZQUEZ, ADRIAN CEDERBERG, PETER VOROBIEFF, University of New Mexico, CLIFFORD HO, Sandia National Laboratories — Particle Image Velocimetry (PIV) is commonly used to extract velocity from a flow field. While these assessments are usually performed using high-speed visible cameras with tracers, experiments performed at the University of New Mexico generate extensive sets of time-resolved thermograms of a falling, hot particle curtain from 400 Hz to 300 Hz. These sets of data can be processed with two commonly used PIV algorithms: DaVis and PIVLab. The comparison showed consistent velocity fields and contours, along with corresponding velocity correlations as a function of flow position. As expected, the vertical velocity component of these gravity-driven curtains follows a trend that resembles a free-falling sphere rather than a falling sphere experiencing drag. The variation of velocity magnitude displayed negligible variations due to the curtain thickness and/or inlet particle temperature which can be considered negligible for the application. The results obtained will feed the development of a statistical model to estimate the mass flow of a particle curtain using only image-correlation methods.

1This research is partially supported by Presto, JST (JPMJPR1678).

Velocity Vector Field Extraction from High Speed Thermograms through Particle Image Velocimetry Tools1. GUILLERMO ANAYA, JESUS ORTEGA, IRMA VAZQUEZ, ADRIAN CEDERBERG, PETER VOROBIEFF, University of New Mexico, CLIFFORD HO, Sandia National Laboratories — Particle Image Velocimetry (PIV) is commonly used to extract velocity from a flow field. While these assessments are usually performed using high-speed visible cameras with tracers, experiments performed at the University of New Mexico generate extensive sets of time-resolved thermograms of a falling, hot particle curtain from 400 Hz to 300 Hz. These sets of data can be processed with two commonly used PIV algorithms: DaVis and PIVLab. The comparison showed consistent velocity fields and contours, along with corresponding velocity correlations as a function of flow position. As expected, the vertical velocity component of these gravity-driven curtains follows a trend that resembles a free-falling sphere rather than a falling sphere experiencing drag. The variation of velocity magnitude displayed negligible variations due to the curtain thickness and/or inlet particle temperature which can be considered negligible for the application. The results obtained will feed the development of a statistical model to estimate the mass flow of a particle curtain using only image-correlation methods.

1This work was funded by the U.S. Department of Energys Solar Energy Technologies Office under the Gen 3 CSP program. Sandia National Laboratories is a multimission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energys Nuclear Security Administration under contract DE-NA0003525.

Experimental study on flow characteristics of vertical upward annular flow boiling in an annulus. JOSEPH SEO, SAYA LEE, DANIEL WACKER, JUNHO LEE, YASSIN HASSAN, Texas AM University — Hydraulic parameters such as outer and inner film thickness, entrainment rate, droplet deposition rate, and droplet velocity of vertical upward annular flow boiling in an annulus have been measured and studied in the present study. The experiment is performed by observing the annular flow boiling in a vertical annular tube by evaporating refrigerant (hydrofluoroether-7000) with a central heating rod. The outer film thickness is measured using planar laser induced fluorescence (PLIF) with Rhodamine-6G as a fluorescence dye while thickness of inner film which is formed at the surface of the heater is visualized and captured by shadowgraph method. The entrainment rate and deposition rate of droplets are also calculated from the image obtained by shadowgraph. Four high-speed cameras with 3000 fps of frame rate are used. The result of the measurement shows that entrainment and deposition rate of droplets is highly related with not only the wave characteristics on the film but also the onset of burn up at the surface of heating rod which are similar with the results from the studies on annular flow. The set of measurement presented in this study is expected to provide a better understanding and insight of the annular flow boiling phenomena in annuli.

A combined structured planar laser-induced fluorescence (S-PLIF) and particle image velocimetry (S-PIV) method for interfacial and near-wall measurements1. VICTOR VOULGAROPOULOS, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — The application of experimental laser-based methods to obtain spatiotemporally resolved information has been emerging as increasingly important in various fields of fluid dynamics, owing to their ability to provide a holistic picture of the phenomena involved and to play a crucial role in benchmarking numerical models. Two-phase systems consisting of fluids with strong variations in their refractive indices in the visible wavelengths, e.g., any gas-liquid system, have, however, been found to compromise traditional laser-based techniques. These optical techniques become susceptible to laser-light reflections and reflections close to the fluid interface, resulting in erroneous interface location and velocity measurements. In this work, we investigate horizontal gas-liquid stratified pipe flows employing a new structured illumination technique. We perform simultaneous structured planar laser-induced fluorescence (S-PLIF) and structured particle image velocimetry (S-PIV) measurements to obtain phase and velocity fields, by minimising total internal reflections effects encountered in traditional PLIF and PIV measurements. The wave characteristics as well as the near-wall and interfacial flow properties are discussed.

1Support from TOTAL and the Transient Multiphase Flows Consortium is gratefully acknowledged.

Pressure Field Estimation on Flow over a Sidewall Aneurysm. PAULO YU, VIBHAV DURGESH, University of Idaho — One of the challenges associated with the experimental fluid dynamic study of flow in an aneurysm is the inability to accurately estimate pressure distribution inside the aneurysm sac. The objective of this study is to estimate the pressure field in the sac using velocity data from Particle Image Velocimetry (PIV) measurements. An in-house experimental setup was developed and an idealized sidewall aneurysm was used for this investigation. A computer-controlled pump system was used to precisely control inflow conditions such as Reynolds number (Re) and Womersley number, which ranged from 50-250 and 2-5, respectively. PIV measurements were conducted on a vertical plane inside the aneurysm sac. Proper Orthogonal Decomposition (POD) was used for low-order reconstruction to reduce the noise in the velocity field. The pressure field was estimated numerically using pressure Poisson equation with Neumann boundary conditions. The estimated pressure field in the pipe was compared with a simulated multi-modal solution for a pulsatile flow showing good agreement. The results of the study showed the evolution of the pressure field inside the aneurysm sac and its dependence on change in Reynolds and Womersley numbers.

Feasibility Study on Sparse Processing Particle Image Velocimetry1. NAOKI KANDA, KOKI NANKAI, YUJI SAITO, Tohoku University, TAKU NONOMURA, Tohoku University, Presto, JST, KEISUKE ASA, Tohoku University — The objective of this study is to investigate accuracy and calculation time of the velocity field data estimated from sparse interrogation area of particle images by Kalman filter and PIV. This estimation method is named sparse processing particle image velocimetry (SPPIV). In this study, the PIV measurement is conducted for NACA0015 under the following conditions; the freestream velocity, the angle of attack, the chord and span of this model were 10 m/s, 16 degree, 100 mm and 300 mm, respectively. Ten proper orthogonal decomposition (POD) modes and five interrogation areas were used and the velocity field was estimated by SPPIV. The optimum location of sparse interrogation area is estimated by greedy method. In this experiment, calculation time of estimating velocity field by SPPIV was approximately 1% of conventional PIV method. In the lowest-order POD mode, the estimated POD mode by SPPIV showed approximately same trend as POD mode calculated by Conventional PIV, however POD mode calculated by SPPIV is sometimes quantitatively different from that calculated by conventional PIV. The quality of particle images and the calculation method of optimum location of sparse interrogation areas should be improved for more accuracy in the future.

1This research is partially supported by Presto, JST (JPMJPR1678).
9:29AM Q11.00009 Development of experimental visualization method for unsteady hydrodynamic stress field by using photoelasticity of liquid polymer\(^1\), MASAKAZU MUTO, YOSHIIUKI TAGAWA, Tokyo University of Agriculture and Technology — The experimental visualization for the unsteady hydrodynamic stress field is in demand for the medical field, because the wall shear stress at the inner blood vessel may be the principal cause of the angioopathy. In this study, we develop a visualization method for the hydrodynamic stress field by introducing liquid polymers with photoelastic effects. Photoelasticity is a non-contact optical measurement method based on phase differences obtained by changes in the polarization state of the polymer solution, which results in proportionate stress field values. In this experiment, to capture the photoelasticity phenomena at high frame rates, a high-speed polarization camera which contains the array of micro linear polarizers with four incident angles is used. As a result, we can observe an increase of phase difference of the liquid polymers (e.g. xanthan gum and carbomethyl cellulose) in milli-channel when their flow rate is increased by syringe pump. Furthermore, the sensitivity of phase difference can be changed by the type and the concentration of the polymers. Especially, this method enables us to reveal the area of hydrodynamic stress concentration in the liquid, which other invasive (contact-type) point measurement steady methods may not capture.

\(^{1}\)JSPS KAKENHI Grants (No. 17H03171 and 17H01246)

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q12 Vortex Dynamics and Vortex Flow: General II
303 - Carlo Scalo, Purdue University

7:45AM Q12.00001 Modification of Tip-Vortices using Chevron Wing Tips, ANUSHKA GOYAL, JOVAN NEDIC, McGill University — The aerodynamic performance of chevrons with varying depths, cut directly into the tips of a flat plate with a semi aspect ratio of 3 were investigated using a time resolved six axis force/torque sensor at a Reynolds number of 67,000. Results show that shallower chevrons cut directly into the tips of wings lead to a higher peak \( \frac{C_l}{C_D} \) ratio at an angle of attack of 5\(^\circ\). It is known that the formation of a tip vortex depends on the geometry of the wing tip (Sarpkaya 1983, Giuni and Green 2013). The chevrons plates formed tip vortices that have lower peak tangential velocities and larger core radii as compared to a flat plate, based on measurements taken by using a four sensor hot wire. The tip vortices formed on wing tips with deeper chevrons exhibited a turbulent core, as opposed to those formed on a flat plate. It was also found that deeper chevron plates had an impact on the wandering of the tip vortex.

7:58AM Q12.00002 Wavelength variation in seal whisker geometries and the effect on vortex structure, CHRISTIN T. MURPHY, Naval Undersea Warfare Center Division Newport, KATHLEEN M. LYONS, University of Wisconsin - Madison, WILLIAM A. HADDOCK, Brown University, WILLIAM N. MARTIN, AREN M. HELLUM, Naval Undersea Warfare Center Division Newport, KENNETH S. BREUER, Brown University, JENNIFER A. FRANCK, University of Wisconsin - Madison — Seal whiskers have a unique undulated geometry that affects water flow over the structure and influences downstream shedding. By intensifying and modifying geometric features in whisker models, we can observe their effects more clearly. In a multi-parameter analysis, wavelength is shown to be an important parameter, especially if interacting with other geometry features. This study isolated the effect of wavelength by creating four physical models of different wavelengths but constant streamwise and transverse amplitudes, peak shift, and symmetry. Flow visualization in a water tunnel, in the biologically relevant Reynolds number range of 500-2000, demonstrates the ability of the undulations to enhance the spanwise momentum transport, reduce the recirculation region, and modify the frequency spectra in the recirculation region behind the whisker. To complement the experiments, direct numerical simulations (DNS) at Re 500 are performed on the four models to correlate the flow structure visualization with resulting drag coefficients, root-mean-square lift coefficients and reduced frequencies. Agreement between experiments and simulations isolates the dominant flow structures responsible for shifts in the frequency spectra over the range of wavelengths investigated.

8:11AM Q12.00003 Direct numerical simulation of trefoil knotted vortices, XINRAN ZHAO\(^1\), CARLO SCALO\(^2\), Purdue University — 3D viscous vortex reconnection has been a topic of strong interest for the fluid mechanics community over the past several decades. This paper investigates pre- and post-reconnection dynamics of a trefoil knotted vortex for Reynolds numbers up to Re = 20,000 by means of DNS with adaptive mesh refinement. The compressible Navier-Stokes equations are solved on the block-structured computational domain with compact-finite difference scheme. An overall high-order accuracy in space can be achieved with the combination of high-order compact restriction/prolongation operators. The test refinement function is given by a Coherent-vorticity-perserving (CvP) sensor (Chapelier, Wasistho, and Scalo, 2018. J. Comput. Phys., 359, 164-182) from our previous work. The simulation on the trefoil vortex problem has shown that this sensor is capable of capturing and refining the location where the reconnection occurs and local turbulence is produced. The complete flow evolution is resolved by the DNS simulation, including the turbulence production upon reconnection, subsequent separation into a smaller and a larger vortex ring, and, finally, the formation of Kelvin waves. The DNS simulation depicts the mechanism how helicity is produced due to small-scale vortical events during the bridging process. A qualitative comparison between the present simulations and existing experiments has also been conducted and an excellent match has been found in terms of flow topology.

\(^{1}\)Postdoc fellow
\(^{2}\)Assistant professor

8:24AM Q12.00004 Vortex Force Map Method for Unsteady Incompressible Viscous Flows\(^1\), JUAN LI, XIAOWEI ZHAO, The university of Warwick — Recently, Li & Wu (2017, 2018) proposed a vortex force map method for a flat plate and extended it to general airfoils at high Reynolds numbers by adopting Howe’s (1995) force formula for the derivation of the vortex force vectors. Here, the vortex force map method is refined to have the capability of dealing with more general cases for a larger range of Reynolds numbers. Updated vortex-pressure force maps, which ensure vortices far away from the body have negligible effect on body-force, are built based on the vortex-pressure force factors derived from viscous governing equations. These maps help us identify the force contribution role of each given vortex more precisely than the ones presented previously. The impulsively started flows around a cylinder and a NACA0012 airfoil are used to demonstrate the applications of this vortex force map method. CFD is used to provide the velocity and vorticity data, as well as for validations by comparing the forces directly given by it with those from the proposed vortex force map method. In order to explore the possibility of applying this method to extracting forces from PIV data, the accuracy of this approach on small domains and under coarse sampling is demonstrated.

\(^{1}\)the Marie Sklodowska-Curie grant agreement No.765579
8:37AM Q12.00005 Vortex dynamics from a vibrating leaf upon drop impact, ZIXUAN WU, SEUNGHO KIM, SUNGWHA NUNG, Cornell University — Raindrop impact have been shown to discharge rust spores and induce dispersal from leave surface through fluid-elasticity interactions and vortex ring formations. In contrary to impacts on rigid surface, here we present an exposition on the vortex dynamics transformations resulting from drop-induced leaf oscillations. Experimentally, we utilized high-speed imaging to probe the complex vortex dynamics and (Single)-(Pair) shedding schemes from a damped harmonic oscillator (drop impacted a flexible, free-end beam) without a prescribed background flow. While low impact inertia is shown to yield single vortex shedding behavior, epitomized by the von Karman street, beam fluctuations from high inertia and to longitudinal twisting motions can give ways to paired vortex schemes and atypical vorticity generations. By tuning such interactions between drop inertia and beam elasticity, the beam vibrations show different frequency and amplitude regimes, with damping effect from lingering vortices and residual drop oscillations. Vorticity behavior of the induced airflow can then provide potential insights on more spore dispersal from simple mechanically induced vibrations.

1This work was supported by National Science Foundation Grant CBET-1604424 and US Department of Agriculture Grant 2018-67013-28063.

8:50AM Q12.00006 The Propulsive Performance of Side-By-Side Foils at a Range of Re and St, AHMET GUNGOR, ARMAN HEMMATI, University of Alberta — The hydrodynamic interactions between two foils placed in side-by-side arrangement are investigated using Direct Numerical Simulation at Reynolds numbers of 1000, 2000 and 4000, and Strouhal numbers (St) ranging from 0.25 to 0.5. The transverse spacing and phase difference between the foils are kept constant. Dynamic motions of the foils are simulated using dynamic mesh morphing technique in OpenFOAM. Coefficients of thrust and power, as well as efficiency, are used to compare the performance of foils. The interactions between foils are also studied through varying the St of individual foils. The performance parameters of the system are observed to depend on both St and Re. In the range of St, the combined efficiency of the system reaches the maximum at St=0.4. The system experiences a thrust enhancement with increasing Re although the coefficient of power remains stable in the Re range. The results are used to evaluate the applicability of the scaling law developed by Floryan et al. (2017) on an isolated foil. The scaling laws for tandem foils in side-by-side arrangements are also developed.

9:03AM Q12.00007 ABSTRACT WITHDRAWN —

9:16AM Q12.00008 Drag Estimation of Isolated, Surface-mounted, Droplet-inspired Geometries, XUEQING ZHANG, BURAK A. TUNA, SERHII YARUSEVYCH, SEAN D. PETERSON, University of Waterloo — Droplet mobility on a substrate due to aerodynamic loading is of interest of many industrial applications. However, modeling this phenomenon is hindered by a lack of reliable estimations of aerodynamic forces on representative shapes. The present study investigates the wake development downstream of isolated, surface-mounted, three-dimensional droplet shapes submerged in a laminar boundary layer. The obstacle geometries considered are representative of the droplet morphologies at sessile state (‘sessile’) and at depinning (‘runback’). The incoming flow has a Reynolds number based on obstacle height of $Re = 2070$ and a boundary layer thickness of around one obstacle height, simulating the critical flow conditions at droplet depinning. Aerodynamic loading on the obstacles is estimated using the wake integral method extended to drag estimation for obstacles with high boundary layer submergence. With a laminar incoming boundary layer, the drag coefficient of the ‘sessile’ model is larger than that of the ‘runback’ model. The drag reduction for the ‘runback’ model is ascribed to the tapered front-body and short aft-body geometry, which makes the obstacle more aerodynamic. For both cases, drag coefficients decrease with increasing turbulence level in the incoming flow.

9:29AM Q12.00009 Definition of local vortical axis flow geometry, KATSUYUKI NAKAYAMA, Aichi Institute of Technology — Local vortical axis flow is defined in the vortical region in the velocity gradient field, which specifies the characteristics of the passage of a vortical axis in the core region of a vortex. The axis vector field derived from vortical axes (identified by a definition of the vortical axis) is defined, and the local axis geometry of the axis vector field is specified by the gradient tensor of its axis vector. Even though the eigenvalues of the tensor may exhibit the feature of the axis curve, it is not associated with the characteristics of the passage of the core region of a vortex. The present study specifies the convergence/divergence/rotation of a bundle of the axes in swirl plane of a vortex. The swirlity and sourcity that represent the unidirectionality and intensity of respective azimuthal and radial flows in the plane are applied to the core region of a vortex. The present study specifies the convergence/divergence/rotation of a bundle of the axes in swirl plane of a vortex. The present study specifies the convergence/divergence/rotation of a bundle of the axes in swirl plane of a vortex.

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q13 Convection and Buoyancy-driven Flows: Heat Transfer and Forced Convection 304 - Chenguang Zhang, MIT

7:45AM Q13.00001 Heat transport by baroclinic acoustic streaming, JACQUES ABDULLASSIH, University of New Hampshire, GUILLAUME MICHEL, Ecole Normale Superieure, CNRS, CHRISTOPHER WHITE, GREG CHINI, University of New Hampshire — Recently, Chini et al. [J. Fluid Mech., Vol. 744 (2014)] and Michel & Chini [J. Fluid Mech., Vol. 858 (2019)] demonstrated that strong acoustic streaming flows can be generated in gases subjected to an imposed cross-channel temperature gradient. In contrast with classic Rayleigh streaming, standing acoustic waves of $O(\epsilon)$ amplitude acquire vorticity owing to baroclinic torques acting throughout the domain rather than via viscous torques acting in Stokes boundary layers. More significantly, these baroclinically-driven streaming flows have a magnitude that is $O(\epsilon)$, i.e. comparable to that of the sound waves, leading to fully two-way wave/flow coupling. The present investigation extends these earlier studies by relaxing the restriction to small aspect-ratio domains, thereby enabling the (forced) heat transport across the channel to be quantified as a function of aspect ratio. This extension requires the numerical solution of a two-dimensional eigenvalue problem for the sound-wave frequency and mode structure. Nevertheless, the resulting computations are orders of magnitude faster than DNS of the compressible Navier-Stokes equations. The prospect for using baroclinic acoustic streaming as a cooling technology is evaluated.
7:58AM Q13.00002 Experiments of Flow Boiling of R245fa in a Horizontal Pipe Using Particle Image Velocimetry\textsuperscript{1}.

- Hannah Moran, Victor Voulgaropoulos, Dimitri Zogg, Omar Matar, Christos Markides, Imperial College London — A bespoke facility is used to measure flow boiling of the refrigerant R245fa in a 12.7-mm diameter stainless steel pipe, to which uniform heating of up to 135 kW/m\textsuperscript{2} may be applied. A range of measurement techniques are utilised in the facility to obtain results over a range of flow rates and heat fluxes. Differential pressure measurements are taken, whilst thermocouples at the tube walls allow the calculation of the heat transfer coefficient, and flow visualisation is accomplished using high-speed imaging. The results are then compared to other experimental work and to correlations in the literature. In addition, laser-based measurements, such as particle image velocimetry (PIV), are performed, providing detailed spatially- and temporally-resolved information on the velocity and turbulence characteristics in the liquid phase. The measurements provide new insight into the hydrodynamic and thermal interactions in these flows and help to build a comprehensive picture of the phenomena involved in the boiling process.

\textsuperscript{1}Imperial College London Marit Mohn Scholarship for HM, UK EPSRC [EP/P004709/1], Royal Society-DFID Africa Capacity Building Initiative, RAEng/PETRONAS Research Chair for OKM

8:11AM Q13.00003 Vibratory control of heat transfer in fluid in containers with elastic boundaries\textsuperscript{1}.

- Nikolai Koizlov, Institute of Continuous Media Mechanics UrB RAS — An experimental study is carried on of heat transfer in a viscous fluid enclosed in cylindrical and spherical containers with vibrating elastic boundaries. The container boundary is thermally stabilized by pumping a liquid around it. Two cases are considered: the fluid bulk is heated while the boundary is cooled and the other way round. Vibrations generate steady streaming in the container, intensifying the heat transfer. The former is studied using PIV method, the latter – by evaluation of temperature difference between the fluids in the container and the thermostatic bath. The dimensionless velocity of steady flows increases with the pulsation Reynolds number, Re\textsubscript{p}. The steady flows are in competition with the free convection. The domain of dimensionless parameters (Rayleigh number, Ra, and Re\textsubscript{p}) where the free convection can be neglected is found. For this domain, the analysis of heat transfer by steady streaming as a function of the dimensionless vibration frequency is done. The obtained results can be applied for the assessment of flow structure and heat-mass transfer in oscillating elastic containers of various scale, for example liquid drops.

\textsuperscript{1}The study was supported by the Government of Perm Region (Programs for the support of International Research Teams, grant C-26/174.9, and Scientific Schools of Perm Region, grant C-26/788).

8:24AM Q13.00004 DNS of thermal channel flow for Re\textsubscript{x} = 5000\textsuperscript{1}.

- Francisco Alcantara Avila, Sergio Hoyas Calvo, Universitat Politècnica de València — A new DNS of a thermal channel flow has been conducted at Re\textsubscript{x} = 5000. The thermal field has been considered as a passive scalar and the Mixed Boundary Condition is employed. The Prandtl number of air, 0.71, has been used. A large enough computational box of dimensions 2\textit{h} x 2\textit{h} x 2\textit{h} has been set to obtain consistent statistics. The mesh used had a total of 6144 x 128 x 6144 \approx 5e10 points. The simulation has run on 2048 cores of the supercomputer MareNostrum. The logarithmic region of the mean temperature profile is starting to be properly developed with a von Karman constant of k\textsubscript{v} = 0.444. Maxima of the temperature intensities increase and move towards the wall with respect to other cases with lower Reynolds numbers. A power function has been obtained to calculate the Nusselt number as a function of Reynolds, for Pr = 0.71. Turbulent Prandtl number does not show remarkable differences with the ones obtained for lower Reynolds and it keeps being close to 1 near the wall. Finally, turbulent budgets for heat fluxes, temperature variance and its dissipation rate have been calculated. Scaling of all terms is analysed and discussed.

\textsuperscript{1}This work was partially supported by MINECO/FEDER, under Project No. RTI2018-102253-B-100. Francisco Alcantara Avila is partially funded by GVA/FEDER project ACIF2018. The computations of the new simulations were made possible by a generous grant of computing time from the Barcelona Supercomputing Centre, reference IM-2019-2-0016.

8:37AM Q13.00005 Toroidal-helical pipe as a passive mixing and heat transfer device in laminar flows.

- Chenguang Zhang, Massachusetts Institute of Technology, Krishnaswamy Nandakumar, Louisiana State University & The Energy Research Institute, Jia, PRC — The secondary flow of Dean vortex pair is a long-known mechanism for the enhancement of mixing and heat transfer, with example uses including circular or spiral pipes. We present a novel design of a toroidal helical pipe, which follows a three-dimensional toroidal helix (an analytical curve smoothly winding around a torus) as its pipe axis. Unlike the circle or straight helix which has constant curvatures, the toroidal helix has a spatially oscillating curvature. This unique feature causes the classic separated Dean vortex pair to couple, periodically re-orient, shrink and grow within each turn of the pipe. The complex flow pattern spans the entire cross-section and enhances mixing via advection. Using heat transfer as an example, the Nusselt number in the toroidal helical pipe varies periodically, with mean values several times higher than a straight pipe even at moderate Reynolds numbers.


- Matthias Ziefuss, Amirfarhang Mehdizadeh, Civil and Mechanical Engineering Department, School of Computing and Engineering, University of Missouri-Kansas City — Heat transfer modeling plays a major role in design and optimization of modern and efficient cooling systems. However, currently available models suffer from a fundamental shortcoming: their development is based on the general notion that an accurate prediction of the flow field will guarantee an appropriate prediction of the thermal field, known as the Reynolds Analogy. This analogy works reasonably well when applied to fluids with a Prandtl number around unity to obtain first order statistics. Concerning fluids with non-unity Prandtl number, there is no comprehensive assessment available. Thus, this investigation presents an introductory assessment of the capability of the Reynolds Analogy when applied to turbulent shear flows of fluids with small Prandtl number. The assessment includes steady and unsteady state simulations. In case of steady state simulations, it turns out that the Reynolds Analogy is not able to predict the mean temperature at an acceptable level of accuracy, while second order statistics are severely mispredicted. In case of unsteady simulations, it is shown that the Reynolds Analogy cannot be considered as an appropriate sub-grid scale model as it fails to feature basic properties of a reliable sub-grid scale model.
9:03AM Q13.00007 Investigating the Impact of Variable Properties for Cryogenic Helium in a Heat Transfer in a Microchannel. COHA SHIN, Oregon State University, MICHAEL MARTIN, SHASHANK YELLAPANTULA, MARC HENRY DE FRAHAN, RAY CROUT, National Renewable Energy Laboratory — Cryogenic helium in liquid, vapor, and supercritical phases is used for cooling in a range of applications, including quantum computers, superconducting magnets, and infrared sensors used in astronomical measurements. Properties such as density, viscosity, thermal conductivity, and specific heat vary significantly near the critical point (5.2 K and 227.46 kPa). We have simulated laminar flow in a heated square microchannel with widths ranging from 25 to 100 microns under supercritical conditions using Pecos computational fluid dynamics code with adaptive mesh refinement, embedded boundaries, and complex fluid equations of state. At temperatures between 30 and 60 K, the Soave-Redlich-Kwong equation of state is used to calculate thermodynamic properties. A temperature and pressure sensitive model is also used for transport properties. As the temperature changes due to wall heating, we observe the variations of properties in the fluid due to these models, and measure the impact on flow structure and channel heat transfer.

1National Science Foundation Mathematical Sciences Graduate Internship Program and Department of Energy’s Office of Science through the Exascale Computing Project

9:16AM Q13.00008 Flow field and heat transfer characteristics in a model solar PV farm. JAMES MCNEAL, ANDREW GLICK, NASEEM ALI, JULIAAN BOSSUYT, GERALD RECKTENWALD, Portland State University, Maseeh College Of Engineering, MARC CALAF, The University of Utah, RAL BAYOÁN CAL, Portland State University, Maseeh College Of Engineering, — Large scale solar farms supply an increasing amount of the world’s electricity supply. However, high operation temperatures can strongly reduce efficiency and panel lifetime, negatively affecting the levelized cost of energy. In this work, the convective heat transfer coefficient for a utility-scale solar farm is studied using thermal and particle image velocimetry measurements in a wind tunnel experiment. The results confirm the applicability of the scaled experimental setup to studies of large solar arrays. Further, the velocity measurements indicate the complex flow structure within the solar array, governed by wakes directed upwards due to the orientation of the solar panels.

1This work is funded by the US Department of Energy (DOE) PVRD2 program under award number DE-EE0008168.

9:29AM Q13.00009 Temperature reduction through system level flow enhancement via model solar PV farm wind tunnel experiments. ANDREW GLICK, JULIAAN BOSSUYT, NASEEM ALI, Portland State University, MARC CALAF, University of Utah, RAUL BAYOÁN CAL, Portland State University — To further facilitate the reduction in cost of photovoltaic energy, new approaches to limit module temperature increase in natural ambient conditions should be explored. Thus far only approaches based at the individual panel level have been investigated, while the more complex, systems approach remains unexplored. Here, we perform the first wind tunnel scaled solar farm experiments to investigate the potential for temperature reduction through system level flow enhancement. Results indicate that significant changes in the convective heat transfer coefficient are possible, based on wind direction, wind speed, and module inclination. We show that 30-45% increases in convection are possible through an array-flow informed approach to layout design, leading to a potential overall power increase of ~ 5% and decrease of solar panel degradation by + 0.3%/year. Previous models demonstrating the sensitivity to convection are validated through the wind tunnel results, and a new conceptual framework is provided that can lead to new means for solar PV array optimization.

1The work presented herein were funded by the US Department of Energy (DOE) PVRD2 program under award number DE-EE0008168.

Tuesday, November 26, 2019 7:45AM - 9:42AM — Session Q14 Wind Turbines: Vertical Axis

7:45AM Q14.00001 Blade-Wake Interactions in a Vertical Axis Wind Turbine. SANG-WOO AHNN, Seoul National University, HYEONGMIN KIM, Hyundai Motor Company, SEHYEONG OH, HAECHEON CHO, Seoul National University — We investigate the flow characteristics of a vertical axis wind turbine (VAWT) and its aerodynamic performance using large eddy simulation with an immersed boundary method. The VAWT considered in this study consists of three blades, and the Reynolds number is 80,000 based on the rotor diameter and free-stream velocity. The simulation results show that each blade interacts with the wakes generated by the preceding blade and also by itself in the downwind region. To examine the effect of these blade-wake interactions on the aerodynamic performance, we simulate the flows with one blade only and with two blades, respectively. The blade performance significantly deteriorates in the upwind region due to the wake induced by the preceding blade. In the downwind region where the blade performance is poor, each blade interacts with its own wake and the wake from the preceding blade, but these blade-wake interactions rather improve the aerodynamic performance of each blade. As the tip speed ratio increases, the preceding blade highly influences the performance of the following blade, but does not much on the second following blade.

1This work was supported by the National Research Foundation through the Ministry of Science and ICT (2016R1A1A02921549) and the National Supercomputing Center with supercomputing resources including technical support (KSC-2018-CHA-0030).

7:58AM Q14.00002 A design of active flow control on vertical-axis wind turbines based on resolvent analysis. HSIEH-CHEN TSAI, National Taiwan University — We design an active flow control on vertical-axis wind turbines (VAWT) combining the resolvent analysis and direct numerical simulations (DNS). The immersed boundary projection method is used to simulate the two-dimensional incompressible flow around a NACA0018 three-bladed VAWT at low Reynolds numbers. Localized body forces are placed on the surfaces of the turbine blades to mimic the streamwise plasma jets generated by Dielectric barrier discharge (DBD) actuators. The optimal actuator locations and the optimal actuation frequencies at various Reynolds numbers are determined by the resolvent analysis of the mean flow around the VAWT. Preliminary results show that by removing wake-capturing vortical structures observed in previous studies, the active flow control successively enhances the average torque generated by turbine blades.

1This project is supported by Taiwan Ministry of Science and Technology (MOST 108-2636-E-002-010).
using OpenFOAM, ASMELASH HAFTU, SHIVASUBRAMANIAN GOPALAKRISHNAN, PRABHU RAMACHANDRAN, Indian Institute of Technology Bombay — Vertical Axis Wind Turbines (VAWTs) employ one or more, straight or curved blades which rotate parallel to the axis of rotation. Blade arrangement creates complex aerodynamics and unsteadiness in the flow. The objective of this study is to demonstrate that the model for simulating Darrieus-type vertical-axis wind turbines (VAWTs) using OpenFOAM is accurate and reliable. The study utilizes CFD simulations to characterize the aerodynamic performance of a chord-wise flexible vertical-axis wind turbine (VAWT). Wind-tunnel experiments are performed for rigid and flexible airfoils, over a range of Reynolds numbers, and performance improvements in terms of lift and drag coefficients are discussed. Airfoil simulations are performed using the OpenFOAM framework to help identify any mechanisms of airfoil performance improvement. Finally, airfoil lift and drag data are used with the aforementioned DMST model to investigate any performance improvements in VAWT performance through the use of flexible blades.

8:24AM Q14.00004 2D CFD study of Darrieus type straight single bladed VAWT using OpenFOAM, ASMELASH HAFTU, SHIVASUBRAMANIAN GOPALAKRISHNAN, PRABHU RAMACHANDRAN, Indian Institute of Technology Bombay — Vertical Axis Wind Turbines (VAWTs) employ one or more, straight or curved blades which rotate parallel to the axis of rotation. Blade arrangement creates complex aerodynamics and unsteadiness in the flow. The objective of this study is to demonstrate that the model for simulating Darrieus-type vertical-axis wind turbines (VAWTs) using OpenFOAM is accurate and reliable. The study utilizes CFD simulations to characterize the aerodynamic performance of a chord-wise flexible vertical-axis wind turbine (VAWT). Wind-tunnel experiments are performed for rigid and flexible airfoils, over a range of Reynolds numbers, and performance improvements in terms of lift and drag coefficients are discussed. Airfoil simulations are performed using the OpenFOAM framework to help identify any mechanisms of airfoil performance improvement. Finally, airfoil lift and drag data are used with the aforementioned DMST model to investigate any performance improvements in VAWT performance through the use of flexible blades.

8:37AM Q14.00005 Impact of different strut geometries on the performance of H-Darrieus vertical-axis turbines, THIERRY VILLENEUVE, GUY DUMAS, Laval University — Previous studies have shown that the performance of H-Darrieus vertical-axis turbines are highly sensitive to the struts mounting structure. More precisely, the struts, that are supporting the turbine blades, are detrimental to the power extraction. Indeed, the presence of the struts leads to an additional viscous drag contribution for the moving blades that results in a negative torque contribution at the turbine shaft. In addition to this added drag contribution, the struts also affect the flow field within the turbine, and thus, the forces acting on the turbine blades. In order to better understand these interactions between the blades and the struts, URANS and DDES numerical simulations are conducted for an H-Darrieus vertical-axis turbine with different strut configurations at a high Reynolds number. The results obtained from these simulations show that the struts affect importantly the spanwise distribution of the forces on the turbine blades. Moreover, the numerical simulations provide useful insights into the flow field within the turbine and can help to develop a strut geometry that could extract energy from the flow while minimizing its interference with the blade, and thus, contributing positively to the turbine efficiency.

1The authors acknowledge the Natural Sciences and Engineering Research Council of Canada (NSERC) for their financial support as well as Compute Canada for their supercomputer allocation.

8:50AM Q14.00006 A Wake in the Middle of the Night: 3D-PTV Measurements around Full-Scale Vertical-Axis Wind Turbines, NATHANIEL WEI, IAN BROWNSTEIN, JENNIFER CARDONA, MICHAEL HOWLAND, Mechanical Engineering, Stanford University, JOHN DABIRI, Mechanical Engineering and Civil & Environmental Engineering, Stanford University — Studies of the wake dynamics of wind turbines are critical for the design and optimization of wind farms in order to minimize wake losses and maximize power density. To observe the inherently three-dimensional flow structures in the wakes of full-scale vertical-axis wind turbines (VAWTs), a volumetric particle-tracking velocimetry method was developed for field experiments at the Field Laboratory for Optimized Wind Energy (FLOW) in Lancaster, CA, using six cameras and artificial snow as tracer particles. Velocity and vorticity fields extending up to three turbine diameters into the wake were measured around isolated 2-kW VAWTs: one with five straight blades, and another with three helical blades. Two tip-speed ratios were examined for each turbine. The 3D flow measurements allowed the dynamics of vortical structures in the streamwise, transverse, and wall-normal directions to be analyzed. Additionally, significant differences in wake geometry between the straight- and helical-bladed turbines were observed, which can be explained using a simple vortex-line model. These results help clarify mechanisms responsible for wake recovery in VAWTs, and thus have implications for wind-farm design.

9:03AM Q14.00007 Vertical Axis Wind Turbine Design Using Design-by-Morphing and Bayesian Optimization, HARI MOAZAM SHEIKH, PHILIP S MARCUS, University of California, Berkeley — Vertical Axis Wind Turbines (VAWTs) have not been commercialized due to their low Coefficients of Performance ( Cp) compared to Horizontal Axis Wind Turbines (HAWTs). However, their low Cp’s are likely due to the historical lack of systematic optimization. Recent studies, however, have shown that VAWTs can outperform HAWTs in closely packed arrays in terms of energy produced/area. This observation, combined with their low manufacturing and maintenance costs, and versatility, have sparked renewed interest in VAWTs. In this work, we optimize a VAWT and its concentrator with two techniques: Design by Morphing (DbM), which is a novel design method, and Bayesian optimization. With DbM, the shapes of the VAWT blades and concentrators are determined by morphing baseline shapes together, and the weights of the shapes used in the morph are part of a large Degree of Freedom (DoF) design space. Searching for the optimal design in the high DoF space is not possible using conventional optimization techniques due to the high cost of data for this problem, (i.e., computing or experimentally measuring the Cp of a VAWT). Using Bayesian optimization however, a 6-DoF space is optimized with only 1000 CFD-computed data points and produces an optimized VAWT along with its concentrator.

9:16AM Q14.00008 Vertical Axis Wind Turbine Performance Scaling at High Reynolds Numbers with Varying Solidity, MARK MILLER, Pennsylvania State University, ALEXANDER PIQUE, Princeton University, SUBRAHMANYAM DUVVURI, Indian Institute of Science, MARCUS HULTMARK, Princeton University — The large physical size and numerous design configurations of the Vertical Axis Wind Turbine have made it difficult to fully characterize the operational envelope of these machines. Laboratory experiments have been performed on a limited number of configurations due to the high-cost associated with testing full-scale models in large wind tunnel facilities. Ongoing work at Princeton University and Penn State has aimed to examine VAWT operation in detail using the controlled conditions of the laboratory, while simultaneously matching the full-scale aerodynamic parameters of interest: the Reynolds number and the tip speed ratio. To achieve this, a specialized, high-pressure wind tunnel facility has been rebuilt to recreate the struts also affect the flow field within the turbine, and thus, the forces acting on the turbine blades. In order to better understand these interactions between the blades and the struts, URANS and DDES numerical simulations are conducted for an H-Darrieus vertical-axis turbine with different strut configurations at a high Reynolds number. The results obtained from these simulations show that the struts affect importantly the spanwise distribution of the forces on the turbine blades. Moreover, the numerical simulations provide useful insights into the flow field within the turbine that can help to develop a strut geometry that could extract energy from the flow while minimizing its interference with the blade, and thus, contributing positively to the turbine efficiency.

1The authors acknowledge the Natural Sciences and Engineering Research Council of Canada (NSERC) for their financial support as well as Compute Canada for their supercomputer allocation.
9:29AM Q14.00009 Effect of helical-shape blades on the wake flow characteristics of vertical axis wind turbines. SHUOLIN XIAO, University of Houston, RONGNAN YAO, University of Notre Dame, DANIEL ARAYA, Johns Hopkins University Applied Physics Laboratory, JOHN DABIRI, Stanford University, DI YANG, University of Houston — Vertical axis wind turbine (VAWT) is a widely used type of wind energy harvesting device. In recent years, considerable efforts have been devoted to studying the turbulent wake flow characteristics behind VAWTs. While most previous studies have focused on the VAWT’s with straight blades, limited progress has been made for understanding the wake flow dynamics of VAWTs with helical-shape blades. In this study, the characteristics of turbulent wake flows behind helical-shape VAWTs are investigated both by wind tunnel experiments using the particle image velocimetry (PIV) technique and by large-eddy simulation (LES) using the actuator-line model (ALM). This talk presents the preliminary PIV and LES-ALM results for the wake behind a helical-shape VAWT and compares them to those for a straight-blade VAWT.

1This work was supported by the National Science Foundation Fluid Dynamics program under Grant No 1804214.

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q15 Turbulence: Wall-bounded Flows II 310 - Sheldon Green, University of British Columbia

7:45AM Q15.00001 Statistical State Dynamics Based Study of the Role of Nonlinearity in the Maintenance of Turbulence in Couette Flow1, BRIAN FARRELL, Harvard University, PETROS IOANNOU, MARIOS-ANDREAS NIKOLAIDIS, National and Kapodistrian University of Athens — While linear non-normality underlies the mechanism of energy transfer from the externally driven flow to the perturbation field, nonlinearity is also known to play an essential role in sustaining turbulence. We report a study based on the statistical state dynamics of Couette flow turbulence with the goal of better understanding the role of nonlinearity in sustaining turbulence. The statistical state dynamics implementation used is a closure at second order in a cumulant expansion of the Navier-Stokes equations in which the averaging operator is the streamwise mean. Two fundamentally non-normal mechanisms potentially contributing to maintaining the second cumulant are identified. These are parametric perturbation growth arising from interaction of the perturbations with the fluctuating mean flow and transient growth of perturbations arising from nonlinear interaction between components of the perturbation field. By the method of selectively including these mechanisms in a DNS parametric growth is found to maintain the perturbation field in the turbulent state while the mechanism of transient growth of perturbations arising from scattering by nonlinear interactions suppresses perturbation variance.

1Funded by ERC Coturb Madrid Summer Program and NSF AGS1249929

7:58AM Q15.00002 The Monin-Obukhov length in turbulent Taylor-Couette flow1, PIETER BERGHOUT, ROBERTO VERZICCO, RICHARD STEVENS, DETLEF LOHSE, Physics of Fluids Group and Max Planck Center Twente, MESA+ Institute and J. M. Burgers Centre for Fluid Dynamics, University of Twente, P.O. Box 217., DANIEL CHUNG, Department of Mechanical Engineering, University of Melbourne, Victoria 3010, Australia — Turbulent Taylor–Couette (TC) flow is the shear driven flow in-between two concentric independently rotating cylinders. In recent years, direct numerical simulations and experiments (employing particle imaging velocimetry) revealed the shape of the mean streamwise and angular velocity profiles up to very high Reynolds numbers. However, so far no theory has been able to capture the Reynolds number effects of the mean streamwise velocity profile, and the classical von-Karman logarithmic law only fits in a minimal spatial region. In this talk, we show the application of the Monin-Obukhov length to turbulent TC flow. This length scale delineates the flow regions where the production of turbulent kinetic energy is governed either by shear or by the curvature of streamlines (centrifugal effects). We then derive an equation for the mean streamwise and angular velocity profiles that convincingly collapses the profiles for varying Reynolds numbers. Finally, we extend the analysis to varying radius ratios and find an equally convincing collapse.

1This project is funded by the Priority Programme SPP 1881 Turbulent Superstructures of the Deutsche Forschungsgemeinschaft.

8:11AM Q15.00003 The balance of Reynolds stresses equations in spanwise rotating plane Couette flows at dual states1, ZHENHUA XIA, Zhejiang University — In this work, the terms in the transport equations of the Reynolds stresses are analysed in spanwise rotating plane Couette flows at two different states. Our results show that they are generally of the same shape at two different states, but the state with more roll cells has a larger value. The energy transfer between the secondary and the residual fields shows that the secondary flows are more energetic at the state with more roll cells while the residual field is more energetic in the other state. Furthermore, a local inverse energy cascade is observed in the near wall region at the latter state with less roll cells where the energy is transferred from the residual field to the secondary flow field. Our results support the conjecture that the large-scale secondary flows play a very important role in the dual states of spanwise rotating plane Couette flows.

1the National Science Foundation of China (NSF Grant Nos. 11822208, 11772297)

8:24AM Q15.00004 The moving wall effects on structures in turbulent Couette-Poiseuille flows1, JUN HYUK HWANG, JAE HWA LEE, UNIST — Direct numerical simulation of a turbulent Couette-Poiseuille flow (hereafter, CP-flow) is performed to investigate spatial development of turbulent structures in the asymmetric flows between two parallel planes. The asymmetric CP-flows are generated by imposing the constant moving wall velocity condition on the top wall in the opposite direction to the main flow, and the velocity is varied systematically. As the moving wall velocity increases, the friction Reynolds number on the moving wall increases largely, although it is increased slightly on the stationary wall. Inner-scaled mean velocity profiles show that the logarithmic layer is established clearly on the moving wall, whereas it is shortened on the stationary wall compared to that from turbulent pure Poiseuille flow at similar Reynolds number. Profiles of the turbulent intensities show that the turbulent activity increases/decreases near the moving/stationary wall with an increase of the moving wall velocity. The asymmetric features of the CP-flow are mainly attributed to significant growth of near-wall motions on the top wall throughout the elongated shear layer.

1This research was supported by the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2017R1D1A1A09000537) and the Ministry of Science, ICT & Future Planning (NRF-2017R1A5A1015311).
8:37AM Q15.00005 Large-eddy simulation of Taylor-Couette flow at relatively large Reynolds number , WAN CHENG, King Abdullah University of Science and Technology, DALE PULLIN, California Institute of Technology, RAVI SAMTANEY, King Abdullah University of Science and Technology — We present large-eddy simulations (LES) of the incompressible Navier-Stokes equations for Taylor-Couette flow at relatively high Reynolds numbers. The ratio of the two co-axial cylinder diameters is fixed as $\eta = R_i/R_o = 0.909$ with $R_i$, $R_o$ the inner and outer cylinder radii respectively. The outer cylinder is stationary while the inner cylinder rotates with constant angular velocity $\Omega_i$, leading to the driving Reynolds number $Re_i = (R_o - R_i) R_i \Omega_i / \nu$ with $\nu$ the kinematic viscosity of the Newtonian fluid. Wall-resolved LES is implemented using the stretched-vortex, sub-grid scale model with $Re_i$ in the range $10^5 - 3 \times 10^6$. We develop an empirical flow model for the $Re_{\tau,i} = F(\eta, Re_i)$ relationship where $Re_{\tau,i} = u_{\tau,i} (R_o - R_i)/(2 \nu)$ is the inner-cylinder friction Reynolds number. Comparison of the model behavior with experimental data [van Gils et al., PRL, 106, (2011), van Gils et al., J. Fluid Mech., 706, (2012), Merbold et al., Phys. Rev. E, 87, (2013)] direct numerical simulation [Ostilla-Mónico et al., J. Fluid Mech., 788, (2016)] and the present LES will be discussed.

8:50AM Q15.00006 Direct numerical simulations of a swirling flow in a conical diffuser$^1$, ANKIT AWASTHI, UGO PIOMELLI, Queen's University — We have performed Direct Numerical Simulations of a swirling flow in a conical diffuser using NaSt5000, a spectral-element code. This configuration is a model of the draft tube of a hydroelectric power plant, which is used to increase the flow pressure downstream of the turbine. Separation may adversely impact the performance of the diffuser and should be avoided. A conical diffuser with an opening angle of 20 degrees is chosen. Previous experimental and numerical studies can be used for validation [Clausen et al., Exp. Therm. Fluid Sci., 61(1):39–48, 1993]. The experiment uses a very short inlet section so that at the beginning of the diffuser the flow is not fully developed. The flow is extremely sensitive to the inlet boundary conditions. When the inlet boundary condition for axial and circumferential velocities at the beginning of the diffuser are taken from the experimental study, good agreement is achieved downstream. The use of fully developed pipe flow, on the other hand, results in a very different flow field. The effects of synthetic perturbation and forcing techniques to generate inflow conditions will be described. Future work will include the effect of rough walls on the separation characteristics and pressure recovery.

$^1$NSERC and Hydro Quebec

9:03AM Q15.00007 Is secondary flow of Prandtl’s second kind due to intense Reynolds-stress events$^1$, ATZORI MARCO, RICARDO VINUESA, Linné Flow Centre, KTH Mechanics, ADRIÁN LOZANO-DURÁN, Center for Turbulence Research, Stanford University, PHILIPP SCHLATTER, Linné Flow Centre, KTH Mechanics — We investigate intense Reynolds-stress events in the turbulent flow through ducts square and rectangular cross-sections, with the aim of clarifying their relation with the secondary flow of Prandtl’s second kind. The intense Reynolds-stress structures are defined as connected regions of the domain that fulfill the condition $|uu’| > Hu’v$, where $u$ and $v$ are the root-mean-square, and $H$ is a scalar threshold. In particular, we focus on the fractional contribution of these events to the mean vertical velocity, $\bar{V}$. The comparison between duct and channel flows unveils that: 1) in the core of the duct, the fractional contribution is in very good agreement with that in the channel, despite the presence of the secondary flow in the duct; 2) in the corner of the duct, the fractional contribution is in good agreement with the channel only in a small region below the corner bisector. Both in the core and in corner of the duct, the behaviour of the fractional contribution as a function of the wall distance is significantly different from that of $\bar{V}$. According to our results, the secondary flow of Prandtl’s second kind is not due to intense Reynolds-stress events.

$^1$The support by SSF (grant number BD15-0082) and by the Knut and Alice Wallenberg Foundation is acknowledged.

9:16AM Q15.00008 Quantitative contribution of laminar, turbulence and secondary flow to velocity and temperature in rhombic ducts, NAOYA FUKUSHIMA, Tokai University — In this study, Direct Numerical Simulation of turbulent flow in rhombic ducts have been carried out to investigate effects of the corner angle on the velocity and temperature distribution in the ducts. Due to anisotropy and inhomogeneity of the Reynolds stresses, secondary flow of the second kind, which goes from the center to the corner of ducts, is induced. The secondary flow affects the velocity and temperature distribution in the ducts and is supposed to enhance momentum and heat transfer. Even around the obtuse corner whose angle is 150°, the secondary flow with about 1.5 % of bulk mean velocity is still induced. The origin of the secondary flow has not been clarified yet. Fukagata, Iwamoto and Kasagi (2002) have theoretically driven the FIK-identity to evaluate quantitative contributions of laminar and turbulence to the friction in turbulent channel. In this study, the FIK-identity has been numerically applied to DNS data in the rhombic ducts to evaluate quantitative contributions of laminar, turbulence and secondary flow to the velocity and temperature distribution. From the results, the effects of the corner angle on these contributions are investigated. Finally, one possible origin of the secondary flow will be suggested.

9:29AM Q15.00009 Analysis of Thermal Boundary-Layer Structure and Scale-Dependence in Transcritical Flows at Turbulent Conditions$^1$, J ACK GUO, Department of Mechanical Engineering, Stanford University, XIANG YANG, Department of Mechanical and Nuclear Engineering, Pennsylvania State University, MATTHIAS IHME, Department of Mechanical Engineering, Stanford University — Previous literature has shown inadequacies of commonly employed scaling transformations to collapse the mean temperature profile for flows with large property gradients. This is particularly relevant for flows with large density gradients that are encountered in transcritical wall-bounded flows. Here, we examine the breakdown of the temperature law of the wall as a function of compressibility via direct numerical simulation (DNS) of transcritical channel flows at turbulent conditions. We propose scaling corrections and suggestions towards the development of more accurate temperature profiles, representing an important step towards reliable predictions of highly compressible turbulent flows. We also analyze and discuss turbulent statistics and budgets related to the temperature transport and heat transfer and provide estimates under which conditions commonly employed transformations remain valid at transcritical flow regimes.

$^1$JG acknowledges support from the Charles H. Kruger Stanford Graduate Fellowship.

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q16 Turbulent Boundary Layers III 4c3 - Ivan Marusic, Melbourne
zones in zero-pressure gradient turbulent boundary layers

within a turbulent boundary layer (Re around $10^5$) of this mechanism. A water tunnel DPIV study has been carried out to analyze the geometry and behavior of the low speed streaks forming to flow control. Upon scale actuation, the reversed flow would be prevented from moving further upstream and maintaining attached flow over streaks, of a turbulent boundary layer experiencing an adverse pressure gradient, is the main mechanism that induces scale actuation leading the passive bristling of mako denticles by reversing flow close to the surface. It is hypothesized that the reversing flow within the low speed separation control has been observed over samples of flank skin from a shortfin mako shark. It is hypothesized that this control is enabled by some UMZs with a streamwise extent of 2 boundary layer thickness, a spanwise extent of 0.1 boundary layer thickness, and surface conditions from hydraulically smooth to fully rough. In the logarithmic region, the UMZs exhibit universal behavior irrespective of Reynolds number and surface conditions. The velocity difference across the shear interfaces between UMZs scales with the friction velocity $u_*$ and the wall-normal thickness of UMZs scales with the wall-normal distance. Further, the UMZ statistics provide a direct link between the spatial organization of boundary layer turbulence and the hypothesized attached eddies used to derive velocity statistics in the logarithmic region. The observed universal behavior of the UMZs can also be used to develop and refine the representative eddies used in reduced-order models of high-Reynolds-number boundary layers.

The authors acknowledge the support of The Australian Research Council through a Discovery Grant and the European Research Council through the advanced grant COTURB.

8:11AM Q16.00003 Universality of uniform momentum zones in high-Reynolds-number boundary layers

8:24AM Q16.00004 Observing Reversing Flow in Low Speed Streaks of a Separating Turbulent Boundary Layer

8:37AM Q16.00005 Low speed streaks as triggers for passive bristling of shark scales for turbulent boundary layer separation control

1 The Office of Naval Research (Global) NICOP N62909-15-1-2044

1 NSF REU Grant EEC 1659710, Army Research Office Grant W911NF-15-1-0556
Avalanche of drag-inducing near-wall vortices due to streak transient growth in turbulent channel flow. ERIC STOUT, Texas Tech University, XUERUI MAO, University of Nottingham, FAZLE HUSSAIN, Texas Tech University — A key question in turbulent boundary layers is the evolutionary dynamics of the coupling of near-wall streamwise vortices and streaks. A single low-speed streak in a large ($L^+ = 900$) channel flow is triggered with the typical spanwise perturbation to excite streak transient growth (STG) at Re$_t = 220$, and reveals an avalanche of streamwise vortices, as well as very long structures. Development of hairpin vortices, hook vortices, arched vortices, and very long streamwise vortices (VLSM, much longer than $s^+ = 300$) reveal the avalanche dynamics involved in the spread and development of the near-wall structures. Lift up of near-wall fluid by the streamwise vortices generate new streaks and hence STG, revealing a sequence of dynamical events involving complex interactions between streaks and streamwise vortices, undiscovered in previous studies. The interaction between streamwise vortices and the quiescent regions aid in understanding the growth of near-wall dynamics, even of turbulent spots. Visualization reveals evolution of very long streamwise vortices — a promising finding that may reveal the enigmatic genesis and dynamics of VLSM in turbulent boundary layers.

Dissipation events in wall turbulence. M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University, OLIVER T. SCHMIDT, University of California San Diego — The intermittent nature of turbulent flows is characterized by periods of low intensity that are interrupted by brief extreme events during which quantities such as production or dissipation of fluctuation kinetic energy develop marked peaks. Our study examines the mechanisms of extreme dissipation events in turbulent boundary layers at moderate Reynolds numbers. Conditional sampling is applied to time-series data generated in direct numerical simulations. The results point to a connection between localized dissipation maxima and the formation of hairpin vortices via exponential instability. Specifically, time-resolved conditionally averaged velocity fields at a dissipation event show the characteristic spatial structure of an instability of varicose type, as predicted in linear analyses (M.J.P. Hack & P. Moin, J. Fluid Mech., vol. 844, 2018). Visualizations of vortex-identification criteria recover a hairpin-type structure which coincides with the region of highest dissipation. The analysis identifies the precursors of the dissipation events as perturbations in the streamwise velocity component which give rise to the varicose instability by locally augmenting the shear.

A DNS study of a shear-driven three-dimensional turbulent boundary layer with emphasis on momentum transport. HIROYUKI ABE, Japan Aerospace Exploration Agency — DNS is used to examine a non-equilibrium three-dimensional turbulent boundary layer (3DTBL) over a flat plate owing to sudden imposition of surface spanwise velocity $W_s$. Particular attention is given to the effects of crossflow and Reynolds number on momentum transport. In the simulations, three values of inlet momentum thickness Reynolds number (=300, 600 and 900) are used with several values of $W_s$. The present largest $W_s$ is twice the freestream velocity $U_0$, comparable to the maximum value in the spinning cylinder experiment by Loehmann (1976). After imposing $W_s$, the secondary Reynolds shear stress builds up and the mean spanwise velocity (crossflow) increasingly propagates to the outer region where there is a mean streamwise velocity deficit due to the skewed near-wall Reynolds stresses. As $Re$ increases, the inner region of a near equilibrium 3DTBL becomes increasingly enlarged where the structure parameter is smaller than 0.15. The mean velocity magnitude also exhibits a departure from the classical log law (i.e., a larger $K$ than in a 2DTBL). After turning off $W_s$, the recovery to a 2DTBL is slow in the outer region since the 3D effect persists there.

Direct numerical simulation of a turbulent thermal boundary layer spatially evolving on an isothermal wall from a fully turbulent adiabatic flow. MATTEO GELAIN, Safran Aircraft Engines; Laboratoire EM2C (CNRS - CentraleSupelec); OLIVIER GICQUEL, Laboratoire EM2C (CNRS - CentraleSupelec); ALEXANDRE COUILLEAUX, Safran Aircraft Engines, RONAN VICQUELIN, Laboratoire EM2C (CNRS - CentraleSupelec) — A direct numerical simulation of a spatially evolving turbulent thermal boundary layer is performed in a channel flow at $Re_t = 395$. The domain is made of two parts in the streamwise direction. Upstream, the flow is turbulent, homogeneous in temperature and the channel walls are adiabatic. The inflow conditions are extracted from a recycling plane located further downstream so that a fully developed turbulent adiabatic flow reaches the second part. In the domain located downstream, isothermal boundary conditions are prescribed at the walls. The boundary layer, initially at equilibrium, is perturbed by the abrupt change of boundary conditions and a non-equilibrium transient phase is observed until, further downstream, the flow reaches a new equilibrium state. Near-wall Reynolds stresses increase with crossflow due to the increased straining. As $Re$ increases, the inner region of a near equilibrium 3DTBL becomes increasingly enlarged where the structure parameter is smaller than 0.15. Mean and root-mean-square profiles of temperature and velocity are presented along with budgets of first- and second-order moments balance equations for the enthalpy and momentum turbulent fields.

Tuesday, November 26, 2019 7:45AM - 9:29AM – Session Q17 Focus Session: Recent Advances in Data-driven and Machine Learning Methods for Turbulent Flows

Towards (Machine) Learning of Large Eddy Lagrangian Models (of Turbulence). MICHAEL CHERTKOV, MIKHAIL STEPANOV, University of Arizona — Aimed at developing a physics-informed simulation approach compatible with modern Machine Learning, we focus here on designing, analyzing and experimenting with reduced Lagrangian, multi-particle models, which are capable of capturing fatefully turbulent dynamics within the resolved large-scale portion of the inertial range. We generalize over popular particle-based models, e.g. Molecular Dynamics (MD) and Smooth Particle Hydrodynamics (SPH), known to generate hydrodynamic behaviour at the scales (typically much) larger than the mean-particle distance. The generalization, reflected in introducing sufficient number of interpretable parameters, is inclusive: we allow variability in the choice of (a) MD pair-wise potential, (b) SPH weighting function, and (c) thermodynamic relation between pressure and density. To mimic effects of the under-resolved scales, we include in the model additional regularizations, such as dependence of the potential and of the weighting function on the inter-particle velocity. In order to generate homogeneous, isotropic and weakly-compressible turbulence we force the Lagrangian system at large scales. To gain a qualitative understanding of what can be achieved at large scales we test the model in different regimes.

This work is a part of Machine Learning for Turbulence project funded by LDRD office at LANL/DOE.
7:58AM Q17.00002 Dynamical System Analysis of Data-Driven Turbulence Models

SALAR TAGHIZADEH, FREDDIE WITHERDEN, SHARATH GIRIMAJI, Texas A&M University — Recent advances in machine learning (ML) algorithms, in conjunction with the availability of direct numerical simulation data, have resulted in a surge of interest in data-driven turbulence modelling. The idea with such models is to replace one or more components of a classical closure model with an implicit function obtained through a trained ML procedure. In training such a procedure, data is used to infer unknown turbulence constitutive relationships. However, choices of these learnable functions and the set of modelled equations can be internally inconsistent. Specifically, attributes such as fixed point behaviour, realizability, and consistency with other physical and mathematical constraints such as the rapid distortion limit may no longer be preserved. In this work, we introduce a novel procedure based on fixed point analysis for ensuring that the overall set of equations in data-driven turbulence modelling form a self-consistent dynamical system. The procedure will be showcased on a new data-driven Reynolds averaged Navier–Stokes model which we have developed.

8:11AM Q17.00003 Uncertainty quantification and optimization of spray break-up submodel using regularized multi-task neural nets.

XIANG GAO, Microsoft Research, HONGYUAN ZHANG, KRISHNA BAVANDLA, PING YI, SUO YANG, University of Minnesota — For a high-fidelity simulation of engine combustion, parameters of a spray atomization break-up submodel needs to be optimized for the specified conditions to match with the non-reactive experiment. The well-accepted KH-RT spray breakup model include at least 6 parameters and they are not independent of each other, thus cannot be optimized independently. Properly tuning is time-consuming and often need expertise-guide. We propose a regularized multi-task neural nets approach to find optimal submodel parameters \( \theta \) at the working condition \( X \) that minimizes “error” \( \epsilon \). The proposed model includes two neural nets: a predictor and an autoencoder. Predictor is trained to predict the submodel parameters \( \theta \) for a given \( X \) and \( \epsilon \). The optimal \( \theta \) then can be estimated by setting \( \epsilon \) as zero. Autoencoder is used to learn a latent representation of a pair of \( (X, \theta) \), which is encouraged by a regularization term to share the same latent space as the predictor. For an unseen condition \( X \) and estimated optimal \( \epsilon \), we can use the autoencoder to find similar \( (X, \theta) \) pairs from the training data to interpret the predictor prediction and quantify the uncertainty.

8:24AM Q17.00004 Approximate Bayesian Computation for Parameter Estimation in RANS Turbulence Models

OLGA DORONINA, University of Colorado, Boulder, SCOTT MURMAN, NASA Ames Research Center, PETER HAMLINGTON, University of Colorado, Boulder — Traditionally, turbulence model parameters have been determined through either direct inversion of model equations given some reference data or using optimization techniques. However, the former approach becomes complicated for models with many different parameters or when the model consists of partial differential equations. Here, we use an Approximate Bayesian Computation (ABC) approach to estimate unknown model parameter values, as well as their uncertainties, in a nonequilibrium anisotropy closure for Reynolds averaged Navier-Stokes (RANS) simulations. ABC does not require direct computation of a likelihood function, thereby enabling substantially faster estimation of unknown parameters as compared to full Bayesian analyses. Details of the ABC approach are described, including the use of a Markov chain Monte Carlo technique as well as the choice of summary statistics and distance function. Unknown model parameters are estimated based on reference data for different homogeneous nonequilibrium test cases. We also discuss the calibration of turbulence models in inhomogeneous flows using forward simulations of an axisymmetric bump.

8:37AM Q17.00005 Machine-learning-assisting investigation of turbulence anisotropy

JUNYI MI, CHAO JIANG, SHUJIN LAIMA, HUI LI, Harbin Institute of Technology — An anisotropy invariant map (AIM) and barycentric map (BMap), which are based the space spanned by invariants of the anisotropic stress tensor, have been playing a crucial role in stress invariant analysis to quantify turbulence anisotropy. However, these methods cannot offer any scale information about the turbulent structures, otherwise the degree of axisymmetry and anisotropy. Only a mingy portion of the turbulent flow in real world can reach the edges or vertices of the AIM or BMap, with which therefore a deeper understanding of flow details about the flow pattern regimes is rarely developed. Here we report an unsupervised machine-learning algorithm (a modified K-means method) as a classifier of flow pattern regimes, with the Reynolds stress tensor instead of their secondary quantities as input features (including the distances from the walls). Tests are performed in duct flows. As a result, (i) there is a consistent one-to-one match between the separation boundaries for different regimes in flow space and the border of invariants for this flow itself in the AIM or BMap; (ii) the size of flow regime in coordinate space leads to identifying the scales of turbulent structures. Besides, effects of the aspect ratio and Reynolds number are examined.

8:50AM Q17.00006 Predicting the stochasticity: GAN-VAE based deep learning model for turbulence prediction

CHANGLIN JIANG, AMIR BARATI FARIMANI, Carnegie Mellon University — Turbulence is a classical tempo-spatial system that has high non-linearity and prohibitively large degrees of freedom, which is basically considered impossible to solve analytically. Unlike Computational Fluid Dynamics (CFD) techniques which generally have considerable time and memory consumption, deep learning (DL) models could learn the hidden features automatically and hierarchically in multiple levels given massive dataset. Instead of solely learning to do turbulence parameterization, we seek to predict turbulence without any underlying physical rules. Our GAN-VAE based DL model could successfully simulate the ambiguous nature of turbulence by combining two distinct but complementary approaches: (a) a variational auto-encoder (VAE) which explicitly models stochasticity by latent layer sampling. (b) a generative adversarial network (GAN) that aims to produce realistic predictions. In the meantime, our model takes advantage of recent work in object detection and motion prediction, such as Convolutional Dynamic Neural Advection (CDNA) and convolutional LSTM. The model prediction is assessed with physics-based metrics like small scale statistics and flow morphology. Our results show promising capability of predicting turbulence that satisfies physical rules with high accuracy.

9:03AM Q17.00007 Using machine learning to predict low-altitude atmospheric optical turbulence

CHRIS JELLEN, JOHN BURKHARDT, CHARLES NELSON, CODY BROWNELL, U.S. Naval Academy — Laser-based systems employed within the atmosphere under layer are subject to degradation due to index of refraction fluctuations within the beam path, known as optical turbulence. Laser propagation through optical turbulence results in beam spread, loss of coherence, and reduced irradiance on target. The root causes of optical turbulence are temperature and humidity fluctuations within the atmosphere. Prediction of these parameters from basic meteorological data is required for effective implementation of any long-range laser system. A field measurement site at the U.S. Naval Academy in Annapolis, Maryland, is used to gather data related to atmospheric effects on optical propagation. A scintillimeter measures the refractive index structure constant along a 1-km path over the Severn River adjacent to the Chesapeake Bay. At each end of the scintillometer link, weather data including wind velocity, air and sea surface temperatures, etc. is captured. Using this data, machine learning techniques are used to predict the refractive index structure parameter of the atmosphere and the scintillation on target of the laser.

1This study is financially supported by the National Natural Sciences Foundation of China (NSFC) under grant Nos. U1711265 and 51505138. We wish to thank Prof. R. Vinnemes for providing their DNS data of duct flows.

9:43AM Q17.00008 Turbulence Modeling with Deep Learning

PETER HAMLINGTON, University of Colorado, Boulder, SCOTT MURMAN, NASA Ames Research Center, PETER HAMLINGTON, University of Colorado, Boulder — Traditionally, turbulence model parameters have been determined through either direct inversion of model equations given some reference data or using optimization techniques. However, the former approach becomes complicated for models with many different parameters or when the model consists of partial differential equations. Here, we use an Approximate Bayesian Computation (ABC) approach to estimate unknown model parameter values, as well as their uncertainties, in a nonequilibrium anisotropy closure for Reynolds averaged Navier-Stokes (RANS) simulations. ABC does not require direct computation of a likelihood function, thereby enabling substantially faster estimation of unknown parameters as compared to full Bayesian analyses. Details of the ABC approach are described, including the use of a Markov chain Monte Carlo technique as well as the choice of summary statistics and distance function. Unknown model parameters are estimated based on reference data for different homogeneous nonequilibrium test cases. We also discuss the calibration of turbulence models in inhomogeneous flows using forward simulations of an axisymmetric bump.

This study is financially supported by the National Natural Sciences Foundation of China (NSFC) under grant Nos. U1711265 and 51505138. We wish to thank Prof. R. Vinnemes for providing their DNS data of duct flows.

Gratefully acknowledge support from ONR and DE-JTO.
9:16AM Q17.00008 Convolutional Neural Networks for the Solution of the 2D Poisson Equation with Arbitrary Dirichlet Boundary Conditions, Mesh Sizes and Grid Spacings1

ALI GIRAYHAN OZBAY, Department of Aeronautics, Imperial College London, PANAGIOTIS TZIRAKIS, GEORGIOS RIZOS, BJORN SCHULLER, Department of Computing, Imperial College London, SYLVAIN LAIZET, Department of Aeronautics, Imperial College London — The Poisson equation is a problem commonly encountered in engineering, including in computational fluid dynamics where it is needed to compute corrections to the pressure field. However, solving the Poisson equation numerically can be very costly, especially for large-scale problems. We propose a fully convolutional neural network (CNN) architecture to infer the solution of the Poisson equation on a Cartesian grid of arbitrary size and grid spacing, given the right hand side term, Dirichlet boundary conditions and grid parameters. Analytical test cases indicate that our CNN architecture is capable of predicting the correct solution of a Poisson equation with mean percentage errors of a few percentage points and a reduction in wall-clock time compared to traditional solvers based on finite difference methods.

1EPSRC UKTC (Project Ref EP/R029326/1) and Department of Aeronautics, Imperial College London

Tuesday, November 26, 2019 7:45AM - 9:29AM –
Session Q18 Compressible Turbulence 400 - Ivan Bermejo-Moreno, USC

7:45AM Q18.00001 Compressibility Effects in High Speed Turbulent Shear Layers Revisited1

KRISTEN MATSUNO2, SANJIVA LELE, Stanford University — In the past few decades, several models have been proposed to capture the consequences of compressibility on turbulence in shear flows. However, current explanations of reduced growth rates and alterations to turbulence structure with increasing Mach number remain somewhat incomplete, and comprehensive theory and modeling is elusive. In this work, compressible mixing layers over a range of convective Mach numbers (\(M_c \in [0.2, 2.0]\)) and free-stream density ratios (\(\frac{\rho_2}{\rho_1} \in [2, 1.7]\)) are directly simulated and compared to previous work. Using this database, the effect of \(M_c\) on key mechanisms in the evolution of turbulent kinetic energy (TKE) such as the pressure-dilatation correlation and the baroclinic work term are presented. The well-known effects of increasing compressibility on turbulent length scales and anisotropy are also demonstrated. Fluctuating velocity fields are decomposed into solenoidal and dilatational components via a Helmholtz decomposition, and resulting Reynolds stress components display cancellations in the transverse direction. Pressure fluctuations are analyzed to characterize acoustic communication across vortical structures.

1This work is supported by the Department of Energy, Office of Science’s INCITE program

7:58AM Q18.00002 Investigating the dynamics of vorticity and strain rate in compressible turbulent flows

NISHANT PARASHAR, SAWAN S SINHA, Indian Institute of Technology, Delhi, MOHAMMAD DANISH, Bennett University, Noida, BALAJI SRINIVASAN, Indian Institute of Technology, Madras — We examine the effect of compressibility on vorticity and strain rate dynamics for compressible turbulent flows. For this, we employ direct numerical simulations of decaying compressible isotropic turbulent flows. A Lagrangian particle tracker is used to identify the influence of compressibility on vorticity-strain rate dynamics. Time correlations between the instantaneous vorticity vector and the strain-rate eigenvector system are calculated using the Lagrangian history of fluid particles. We show that while the statistics obtained are independent of turbulent Mach number, they are found to be significantly influenced by the locally normalized dilatation rate. Further, we study the time correlations conditioned on local flow topologies (based on invariants of the velocity gradient tensor) as well. We find that the influence of dilatation rate is predominantly associated with rotation dominated flow topologies (unstable-focus-compressing and stable-focus-stretching). At last, we provide a physical explanation of all these observations by tracking the moment of inertia and angular momentum history of tetrahedral fluid elements.

8:11AM Q18.00003 Influence of compressibility on lifetimes of topologies in turbulent flows

SAWAN SINHA, Indian Institute of Technology Delhi, NISHANT PARASHAR, Indian Institute of Technology Delhi INDIA, BALAJI SRINIVASAN, Indian Institute of Technology Madras INDIA — Flow field topologies are categorized based on the nature of eigenvalues of the local velocity gradient tensor. Physically, these topologies are suggestive of the relative importance of the local strain-rate and the rotation rate-tensors. The question that how long a given topology lasts in a turbulent flow field is of fundamental importance in geophysical and astrophysical flows. It has been reported in literature that while in the former, this quest is linked to the process of raindrop formation, in the latter, the question finds its significance in context of star formations. While some earlier attempts have been made to estimate the lifetimes of topologies using surrogate methods like the conditional mean trajectories (CMT), in this work we take a more direct approach and accurately estimate topology lifetimes using well resolved direct numerical simulation data in conjunction with a Lagrangian particle tracker. In particular, we investigate and explain how initial turbulent Mach number and local dilatation rate tends to influence the lifetimes of various topologies in a compressible turbulent flow field. Finally, some modeling implications of these findings are also presented.

8:24AM Q18.00004 Multiscale analysis of passive scalar transfer in compressible isotropic turbulence1

JIANCHUN WANG, MINPING WAN, CHENYUE XIE, QINMIN ZHENG, XIAONING WANG, JIAN TENG, LIAN-PING WANG, SHIYI CHEN, Southern University of Science and Technology, Shenzhen, Guangdong 518055, P. R. China — Inter-scale transfer of a passive scalar in stationary compressible isotropic turbulence is studied by numerical simulations at a Taylor Reynolds number of approximately 250, and at turbulent Mach numbers of 0.4 and 1.0. The 5/3 scaling behavior is identified for the fluctuation spectrum of passive scalar, with the Obukhov-Corrsin constant close to that of a passive scalar spectrum in incompressible turbulence. The average subgrid-scale (SGS) flux of passive scalar normalized by the total dissipation rate is close to 1 in the inertial range. It is shown by Helmholtz decomposition that the SGS flux of passive scalar is dominated by the solenoidal mode of velocity field. Moreover, the effect of local compressibility on the SGS flux of passive scalar is investigated by conditional averaging with respect to the filtered velocity divergence. A discrete approximate deconvolution model (DADM) is proposed to reconstruct the SGS flux of passive scalar from the filtered flow fields. Numerical results show that the SGS flux of passive scalar reconstructed by DADM is in good agreement with the real SGS flux of passive scalar.

1This work was supported by NSFC Grants No. 11702127, No. 91752201, and No. 11672123; Grant No. JCYJ20170412151759222 and ZDSYS201802081843517; Grant No. 2016QNRC001.
8:37AM Q18.00005 Solenoidal scaling laws for compressible mixing\textsuperscript{1}. JOHN PANICKACHERIL, JOHN, DIEGO A DONZIS, Texas A & M University, KATEPALLI R SREENIVASAN, New York University — Mixing of passive scalars in compressible turbulence does not obey the same classical Reynolds number scaling as its incompressible counterpart. In this work we first show from a large database of direct numerical simulations that even the solenoidal part of the velocity field fails to follow the classical incompressible scaling when the forcing includes a substantial dilatational component. Though the dilatational effects on the flow remain significant, our main results are that both the solenoidal energy spectrum and the passive scalar spectrum scale assume incompressible forms, and that the scalar gradient aligns with the most compressive eigenvalue of the solenoidal part, provided that only the solenoidal components are used for scaling in a consistent manner. Minor modifications to this result are also pointed out, in particular the interaction of scalar field with the dilatational part of flow field. Two parameters that are found to be important in compressible mixing are the ratio of dilatational to solenoidal rms velocities and the turbulent Mach number, whose role in mixing will also be discussed.

\textsuperscript{1}Support from NSF award 1605914 is gratefully acknowledged.

8:50AM Q18.00006 Reproducing the local characteristics of compressible turbulent flows at a low cost: derivation and application. GUILLAUME BEARDELL, GUILLAUME BLANQUART, California Institute of Technology — When performing Direct Numerical Simulations (DNS) of highly turbulent reacting flows, it is often prohibitively expensive to simulate complete flow geometries. For example, simulations of turbulence-flame interactions usually do not capture the full combustor, and instead focus on a specific portion of the domain, e.g., the region around the flame front. However, by doing so, one misses turbulent kinetic energy injection due to shear by the large scales. In the present work, we include these large-scale contributions, e.g., from experimental data, and we solve for the small-scale components only. The resulting equations are the same as the original compressible Navier-Stokes equations, except for the introduction of additional terms. This approach allows us to achieve high turbulent Reynolds numbers while keeping the computational cost reasonable. We have already applied this strategy to incompressible flows, but not to compressible ones, where special care must be taken regarding the energy equation. Using the finite-difference solver NGA, we apply this framework to simulations of homogeneous turbulence and premixed flames. We provide comparisons with results obtained with other forcing schemes.

9:03AM Q18.00007 The Interaction of a Homogeneous Field of Acoustic Waves with a Shock Wave. YUCHEN LIU, LIAN DUAN, The Ohio State University — Direct numerical simulations (DNS) and linear interaction analysis (LIA) are used to examine the significant flow characteristics associated with a homogeneous field of acoustic waves passing through a nominally normal shock wave. The full-fledged nonlinear simulations and the linear analysis are enabled by a pre-cursor numerical database of boundary-layer acoustic radiation that provides incident acoustic fields with high degree of physical realism and applicability. The research contributes to the fundamental understanding of the interaction of a shock wave with a field of turbulence by characterizing its behaviors in the pure dilatational limit and complements existing studies of shock/turbulence interaction with a vorticity-dominated incident turbulent field.

9:16AM Q18.00008 Tracking of flow structures in shock-turbulence interaction. JONAS BUCHMEIER, University of Southern California, ALEXANDER BUSSMANN, Technical University of Munich, XIANGYU GAO, IVAN BERMEJO-MORENO, University of Southern California — A tracking algorithm is applied to study the time evolution of flow structures, defined as isosurfaces of a field of interest. Correspondences between structures in consecutive time steps are found in a higher dimensional feature space, which includes a geometric signature of the structures, in addition to their spatial information. To allow for larger tracking steps, constraints are used to reject correspondences based on physical realizability. Accepted correspondences are used to dynamically build a graph describing the evolution and interactions of structures over time. Complex split-merge interactions resulting from large tracking steps are handled based on confidence indicators obtained from the search, physical properties and history of the involved structures. The graph is then queried to retrieve confidence information on the evolution of the structure. The algorithm applies to both shock-capturing DNS of shock-turbulence interaction ($Re_{m} = 40; M_{2} = 0.1, 0.4; M = 1.5, 3.0$). Passive scalar isosurfaces with well-defined initial shapes and turbulence structures based on the Q criterion are tracked across the shock, analyzing relations between local geometry and physical quantities mapped on the surfaces.

Tuesday, November 26, 2019 7:45AM - 9:29AM — Session Q19 CFD: Immersed Boundary Methods II 401 - Andres Goza, University of Illinois at Urbana-Champaign

7:45AM Q19.00001 A Fully Grid-Line-Based Immersed Boundary Method\textsuperscript{1}. GUANTING SU, Beihang University & University of California, Berkeley, TIANYU PAN, Beihang University, QIUSHI LI, Beihang University & Xihua University — By introducing proper variable reconstruction scheme in the vicinity of immersed boundaries, a group of immersed boundary methods (IBMs), including ghost-cell method and hybrid Cartesian method, showed promising performance when simulating unsteady, incompressible flow around complex boundaries using non-body-conformal orthogonal grids. In this work, we propose a fully grid-line-based IBM with a novel velocity reconstruction scheme which is capable of stably imposing desired linear velocity distribution along single or multiple gridline directions. Pressure boundary condition is inherently implemented with desired velocity distribution imposed. Present method greatly simplifies boundary-related operations by eliminating existing need to search for projections along off-gridline direction (e.g., normal to boundary). And with utilization of standard discretization stencils enabled on boundary-adjacent grid nodes, implicit time advancement of viscous term is straightforward. Flow simulation results are in good agreement with reference data and show that the proposed method retains second-order accuracy of the fractional-step Navier-Stokes solver incorporated.

\textsuperscript{1}The authors acknowledge the support of National Natural Science Foundation of China (Nos. 51636001 and 51706008) and Aeronautics Power Foundation of China (No. 6141B090315).

7:58AM Q19.00002 An Efficient Immersed-Boundary Formulation with Explicit Time Stepping for Incompressible Flows. NOAH OSMAN, University of Illinois at Urbana Champaign, ANDRES GOZA COLLABORATION — One of the most persistent difficulties in numerically simulating incompressible flow problems is the treatment of the stiff diffusive term. In order to maintain stability, this term is traditionally treated using an implicit time-stepping method, necessitating the solution to a large linear system at each time step. A variety of efficient techniques have been developed to solve these linear systems. However, their application in the presence of non-uniform grids and other non-canonical numerical configurations remains a challenge. We consider an alternative approach in which the diffusive term is treated explicitly, with no severe restrictions on the time step size. We present our method in an immersed-boundary formulation to enable efficient simulations of flows involving immersed bodies. To do this, we extend the Runge-Kutta Chebyshev method of Van der Houwen and Sommeijer (1980) to the differential-algebraic setting relevant to the immersed-boundary method. We demonstrate the efficacy of the method through results from canonical 2D flow problems.
8:11AM Q19.00003 Particle-resolved DNS(PR-DNS) to study the bulk settling velocity of poly-dispersed particles, YINUO YAO, OLIVER FRINGER, CRAIG CRIDDLE, Stanford University — A PR-DNS is implemented to investigate poly-dispersed particle hydrodynamics in triply-periodic flow. The particle motion is computed with a direct forcing IBM approach along with the collision model proposed by Biegert et al. (2017). The direct forcing method accurately predicts particle motions with moderate particle Reynolds numbers up to 360. To avoid the small time-step size needed to simulate collisions with the soft-sphere approach, we use the adaptive collision time model proposed by Kempe et al. (2012). Poly-dispersed particles are initialized randomly in two- and three-dimensions and subjected to gravity to settle while interacting with one another. Simulations are then run until the mean settling velocity reaches steady state. This relates the velocity to mean particle Reynolds number. Different poly-disperse particle scenarios are devised that allow for the study of the effect of the particle size distribution while keeping other bulk properties fixed. The results are discussed in the context improving energy efficiency of the fluidized bed reactors in the wastewater treatment, which are characterized by the pressure loss due to motion of poly-dispersed particles in turbulent flow. This work is supported by California Energy Commission (CEC).

8:24AM Q19.00004 Treatment of immersed boundaries for the Vortex Particle-Mesh method, THOMAS GILLIS, GREGOIRE WINCKELMANS, PHILIPPE CHATELAIN, UCLouvain — The Vortex Particle-Mesh (VPM) method combines the advantages of a particle method, i.e. low numerical dissipation and dispersion errors, with those of a Cartesian mesh-based approach: highly efficient Poisson solvers and finite difference stencils. However, the accurate treatment of the immersed boundaries in the VPM-framework is still an open challenge as the formulation of the boundary condition for the vorticity is not as straightforward as in a classical velocity-pressure formulation. This complexity is further increased since the VPM method relies on a non-body-conforming Cartesian mesh; hence the obstacle may intersect the grid at arbitrary locations. The two current state-of-the-art methods are the (iterative) penalization approach and the Boundary Element Method coupled with the VPM approach. The first one, the penalization technique, suffers from a lack of accuracy due to the smearing of the interface, hence offering an unsatisfactory surface treatment. The second one, the Boundary Element Method (BEM) approach suffers from a first order time convergence due to the splitting of the diffusion operation. This presentation focuses on this challenge: the treatment of immersed boundaries in order to accurately predict the interactions between the fluid and these structures.

8:37AM Q19.00005 An Immersed Boundary Method for Shock-Particle Interaction, IMAN BORAZJANI, J. Mike Walker 66 Department of Mechanical Engineering, Texas A&M University — A sharp-interface immersed boundary method is developed to simulate the interaction of solid particles with shocks. The inviscid and viscous fluxes of compressible flow equations in curvilinear coordinates are discretized with a third order weighted essentially non-oscillatory (WENO) and a central scheme, respectively. The equations are advanced in time using a third-order Runge-Kutta method. The sharp interface at the immersed boundaries is maintained by reconstructing the compressible flow variables along the normal direction to the boundary similar to the previous method for incompressible flows. The WENO discretization is reverted to a second order ENO scheme near the immersed boundaries to reduce the stencil size and avoid using the nodes inside the immersed boundary in the discretization. The method is validated against experimental measurements and shown to be second-order accurate in the presence of immersed boundaries. The numerical results capture all of the shock features observed in the experiments and show great agreement with the measurements.

8:50AM Q19.00006 High Order Cut-cell Methods in Multiple Dimensions, PETER BRADY, DANIEL LIVESCU, Los Alamos National Laboratory — Cut-cell methods for unsteady flow problems can greatly simplify the grid generation process and allow for high-fidelity simulations on complex geometries. However, cut-cell methods have been limited to low orders of accuracy. This is driven, largely, by the variety of procedures typically introduced to evaluate derivatives in a stable manner near the highly irregular embedded geometry. The present approach is based on two simple and intuitive design principles. These principles, and an a-priori optimization process, allow for the construction of stable 8th order approximations to elliptic and parabolic problems and stable and conservative 5th order approximations to hyperbolic problems. This is done for both explicit and compact finite differences and is accomplished without any geometric transformations, artificial stabilization or other adhoc in-situ procedures. Test cases with 2-D and 3-D geometries will be discussed.

9:03AM Q19.00007 Advanced immersed boundary method for complex moving/morphing boundaries based on hybrid ghost-cell and virtual cut-cell. KAMAU KINGORA, HAMID SADAT, University of North Texas — The state of art of sharp interface immersed boundary (IB) methods (cut-cell and ghost-cell (GC) methods) today is that they lack generality and are specialized in nature. Consequently, their extension to problems with multiple potentials is not straightforward. We propose an accurate and robust advanced immersed boundary method (AIBM) for simulation of fluid structure interaction (FSI) in complex flow. This novel sharp interface IB method entails addition of a generic virtual force to the underlying governing equation for the purpose of enforcing desired boundary conditions on different field variables like velocity, temperature, chemical species etc. AIBM employs virtual cut-cell techniques which makes AIBM easy to implement and parallelize, and also applicable to multiphase flow on grid with high aspect ratio, unlike GC method which rely on interpolation. Solid part of simulation domain is logically eliminated from computation hence AIBM is capable of simulating internal and/or external flow with large number of cells in the solid region. Fresh-cell problem is tackled by a field extension technique, enabling AIBM to simulate FSI with moving and/or deforming boundary without pressure oscillations and spurious forces. We validated the proposed method by running benchmark cases using our in-house solver, CDFFoam. The results predicted by AIBM are in good agreement with experimental studies.

1California Energy Commission (CEC)
9:16AM Q19.00008 Internal Flow Analysis in an Axial Pump using Large-eddy Simulation with Immersed Boundary Method. DANDAN YANG, University of Minnesota; Tsinghua University, XIANWU LUO, Tsinghua University, LIAN SHEN, University of Minnesota — Axial flow pump is widely used in drainage, irrigation, and water diversion applications. A high-fidelity numerical method is needed to study the flow physics and accurately predict the pump performance. In the present work, the transient flow past an axial pump, comprised of the hub, casing, a rotating impeller and a stationary stator, is simulated by large-eddy simulation with the immersed boundary method using a code developed in-house, which is capable of accurately simulating the turbulent flow with three dimensional complex geometries with stationary and moving boundaries. The accuracy of the present simulation method is verified by comparing the predicted hydraulic performance of the pump with experimental results. Based on the simulation results, the internal flow field in the pump is analyzed by investigating velocity and pressure distributions, and turbulent flow structures. Further, pressure oscillation and hydraulic force in the pump are also predicted. The obtained information is helpful to improve the pump design for better performance.

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q20 Experimental Techniques: Aerodynamics / Wind Tunnel 602 - Joanna Austin, California Institute of Technology

7:45AM Q20.00001 ABSTRACT WITHDRAWN –

7:58AM Q20.00002 Estimation of wall-shear stress and its variation in a unperturbed and perturbed plane wall-jet. SRAVAN ARTHAM, Embry Riddle Aeronautical University, SHIBANI BHATT, Embry-Riddle Aeronautical University, ZHENG ZHANG, EBENEZER GNANAMANICKAM, Embry Riddle Aeronautical University — Artham S. PhD in Aerospace Engineering from the University of Washington, focusing on the interaction of shock waves and turbulence. His research interests include flow control, computational fluid dynamics, and experimental techniques. He is currently working on the numerical simulation of shock wave interactions with turbulence and its effects on aerodynamic performance. His work has been published in several reputable journals and presented at various conferences. His mentor is Prof. Douglas Smith. His work is supported by the Air Force Office of Scientific Research under award number FA9550-16-1-0194 monitored by Dr. Douglas Smith.

8:11AM Q20.00003 Smoke visualization over a spinning cone at angle of attack in flight. ABDULLAH KURAAN, ÖMER SAVAŞ, University of California at Berkeley — Smoke visualization over a spinning truncated cone at angle of attack is carried out to study qualitatively the salient features of the flow field. A twisted wire pair is used for generating equally spaced smoke streaks in vertical planes. Detailed images of flow patterns over a spinning cone with a half angle $\theta_c = 30^\circ$ and a base diameter of 20 cm are captured with a high speed camera. The rotational speed of the cone is set at $\Omega = 5$ rev/s, the axial flow at $U_{inj} = 1$ m/s, and the angles of attack up to 36°. The Reynolds number, based on the cone size, is $\sim 10^4$. The analytic description of the axisymmetric potential flow over a fixed cone at zero incidence is reviewed for comparison of the flow field outside the boundary layer. Good agreement in the streamwise distance from the PWJ exit and in the nozzle slot width. These measurements were then curve-fit with direct numerical simulation based velocity profiles to simultaneously extract the WSS as well as a correction to the wall location. The integral method used a momentum integral approach (Mehdi, F. et al. Exp. Fluids, 2014) where PINI-based velocity profiles of the streamwise and wall-normal velocities are integrated to determine the WSS. A reduction in WSS was observed for all the cases considered. The two techniques are compared and key differences are highlighted.

8:24AM Q20.00004 Nitric Oxide Spectroscopic Measurements of a Shock-Boundary Layer Interaction in Hypervelocity Flow. NELSON YANES, JOANNA AUSTIN, California Institute of Technology — Spatially resolved emission spectra are collected in the post bow shock and reattachment shock region of hypervelocity flow over a double wedge. The Hypervelocity Expansion Tube (HET) is used to generate high Mach number, high enthalpy flow ($M = 7, h_0 = 8$ MJ/kg) over a 30-55 degree double wedge. The NO $\gamma$ band ($\Lambda^2\Sigma^+ \rightarrow X^2\Pi$) emission is measured in the UV range of 210-250 nm. Detector exposures occur at select times throughout the flow development process to study temporal changes in thermal and chemical non-equilibrium. Profiles of vibrational band intensity and spectra as a function of distance are provided. Through a fitting procedure, the experimental spectra are matched with synthetic spectra to obtain an estimate of the excitation temperature of the NO molecule. The result is a temperature profile of the post-shock NO* with downstream distance.

8:37AM Q20.00005 Fiber-Optic Michelson Interferometer System for Shock Speed Measurement in an Expansion Tube. WESLEY YU, JOANNA AUSTIN, California Institute of Technology — Shock speed and time of arrival measurements are necessary to properly characterize the flow in high-enthalpy hypersonic impulse facilities. However, the low density test conditions in expansion tubes present challenges for commonly-used wall-mounted static pressure measurements due to low signal levels relative to the amplitude of stress waves resulting from diaphragm rupture or accelerations from driver operation, and an accurate measurement of shock speed is difficult. A fiber-optic Michelson interferometer architecture is being designed and tested for use in shock speed measurements in the Hypervelocity Expansion Tube (HET) at Caltech. The static and dynamic responses of the interferometer are characterized using a pressure vessel and ultrasonic acoustic beam, respectively. Numerical simulations of the interferometer response to a shock wave at experimental conditions, taking into account the structural noise caused by diaphragm opening, indicate that adequate signal-to-noise ratios may be obtained using this technique. This result is encouraging for the development of optical methods to characterize shock speed in impulse facilities with recoil.

1This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-16-1-0194 monitored by Dr. Douglas Smith.

1Since membership application site is still down, was told to put "Membership Pending" until site is back up
8:50AM Q20.00006 Fan array wind tunnels: turbulence on-demand1. CHRISTOPHER DOUGHERTY, ALEJANDRO STEFAN-ZAVALA, PETER RENN, MORTEZA GHARIB, Caltech — Naturally-occurring winds are distinctly unsteady and non-uniform, particularly in complex urban airscapes. To better understand flight in these contexts, it is imperative to be able to control wind profiles in a predictable, repeatable, and representative manner. This can effectively be accomplished with fan array wind tunnels (FAWT), a modularly built multi-source wind tunnel capable of generating a host of spatiotemporally-varying flows via software interfacing. Utilizing an array of DC-powered off-the-shelf cooling fans (in place of one singular drive section) allows for greater flow control, overall decreased mixing lengths, and comparably large useable test section areas when compared with its effective footprint. Tunnel resolution along with fan responsiveness determines the effectiveness of one of its most salient features: the ability to tailor turbulence around a desired a mean (i.e. turbulence on-demand). This decoupling adds new territories to explore in simulated flight contexts. Periodic and unsteady flows will be highlighted with brief considerations given to the future role of machine learning.

1This research was supported by the Center for Autonomous Systems and Technologies at the California Institute of Technology

9:03AM Q20.00007 Flow shaping in wind tunnels with fan array technology. GUILLAUME CATRY, NICOLAS BOSSON, GESHANTH VISVARATNAM, WindShape, FLAVIO NOCA, HEPIA / HES-SO University of Applied Sciences — In the past hundred years, wind tunnels have been built to study the flow of air around objects. In particular, the geometry of the contraction and the diffuser walls has to be carefully designed in order to achieve flat profiles in the test section and avoid boundary layer separation (both in the contraction and the diffuser). The resulting infrastructure has a large footprint and is generally unmodifiable during the whole the lifetime of the wind tunnel. We have developed a technology to shape the morphology of wind in space and time. It is based on a large number of fans (wind pixels), which are distributed arbitrarily in space and can be modulated individually. In particular, we show how this technology allows the wind profile in a test section to be controllable and does not require any a priori design of complex wind tunnel infrastructure.

9:16AM Q20.00008 A low-turbulence transverse gust generator in a wind tunnel1. DAVID OLSON, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — There exists a broad range of aerodynamic problems where the commonly used steady uniform freestream condition is not appropriate. Airfoil-gust interactions are one such problem, with the transverse gust being particularly difficult to study experimentally. We present a novel transverse gust generator consisting of an actuated array of vortex generators mounted to a wind tunnel’s test section. The primary advantage of the design over existing gust generators is its capability to produce a reasonably-uniform transverse stream without producing turbulence in the freestream. The generator’s design can potentially allow for the time history control of the magnitude, direction, and duration of the gust strength. A simplified model for the performance of the design, and the experimental characterization of the gust generator are discussed.

1This work was supported by ONR grant number N00014-16-1-2760.

9:29AM Q20.00009 Determination of drying and shrinkage characteristics of porous food material in convective drying: Model development and experimental studies. PUNIT SINGH2, PRABAL TALUKDAR2, Indian Institute of Technology, Delhi — Food materials are dried to enhance the shelf life, maintain nutritional value, lower packing cost, and reduce shipping cost. An innovative convective dryer is developed, which is similar to a small scale wind tunnel, to determine the drying and shrinkage characteristics of porous food material. A conjugate 3D numerical model is also developed in a commercial software COMSOL to determine the heat and moisture transfer distribution in the food material. The model considers several important phenomena like surface evaporation, internal evaporation, and shrinkage during the drying process to render high accuracy. Results show that the air temperature has significant effect on both drying and shrinkage characteristics as compared to the airflow velocity. Around 80% of moisture is removed in the first 30% of time and remaining 20% of moisture content is removed in the last 70% of time, because of high moisture gradient present in the beginning of drying. The bulk density of food increases initially, reaches a maximum value and finally decreases in the end of drying process. Numerical studies performed with this model will help to design and develop cold storage, estimate the processing time of various operations like cooling and heating.

1Ph.D Student
2Professor

Tuesday, November 26, 2019 7:45AM - 9:42AM — Session Q21 Drops: Spreading and Wetting I 603 - Sadegh Dabiri, Purdue University

7:45AM Q21.00001 Wetting dynamics on asymmetric microstructured surfaces. SUSUMU YADA, SHERVIN BAGHERI, Department of Mechanics, KTH Royal Institute of Technology, JONAS HANSSON, Division of Micro and Nanosystems, KTH Royal Institute of Technology, MINH DO-QUANG, FREDRIK LUNDELL, Department of Mechanics, KTH Royal Institute of Technology, WOUTER VAN DER WIJNGAART, Division of Micro and Nanosystems, KTH Royal Institute of Technology, GUSTAV AMBERG, Sdertorn University — Microstructured surfaces which are able to control the direction of liquid transport are common in nature for fog/water harvesting, surface lubrication, and self-cleaning, and have been inspiring enormous number of man-made structures. However, the spreading of a liquid on such surfaces have been investigated in the slow spreading regime and a fundamental understanding of the early rapid wetting is lacking. In this work, our experimental and numerical investigations on surfaces with periodic patterns of asymmetric microridges provide detailed illustrations of the rapid droplet spreading over complex surface structures. We show that the surface structures are partly wetted as the air-liquid interface above the contact line makes another contact to the structure downstream and creates a new wetting front, leaving some dry surface behind. Furthermore, we elucidate how different physics play roles in different flow directions. In one direction, the spreading is governed by the friction at the moving contact line and the Young’s force related to the local dynamic contact angle, whereas in the other direction, it is determined by the contact line pinning and the inertia of the droplet. Based on these physical insights, the effect of different surface geometry is discussed.
7:58AM Q21.00002 Wetting dynamics of a droplet on micro-pillar surfaces with radially varying pitch. RAJNEESH BHARDWAJ, MANISH KUMAR, Department of Mechanical Engineering, Indian Institute of Technology Bombay, Mumbai 400076, India, KIRTI CHANDRA SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Sangareddy 502 285, Telangana, India — We experimentally investigate the wetting dynamics of a droplet placed gently on a square-micropillar surface. These pillars are located with a radially varying pitch described by a parabolic equation. Two sets of surfaces with radially increasing and radially decreasing pitches from the center of the substrate at which the droplet is initially placed have been prepared on silicon wafer using photolithography. Due to the radial variation of the pitch, the droplet experiences a wettability gradient (either increasing or decreasing radially). We observed that on the surface with the radially increasing pitch, the droplet remains in the Cassie state and exhibits higher contact angle than the smooth surface during its spreading stage. On the other hand, in case of the surface with radially decreasing pitch, the droplet goes into the Wenzel state and assumes a lower contact angle as compared to that observed in the smooth surface. The wetted diameter of the droplet is found to be smaller in case of the radially decreasing surface than the radially increasing surface. We also studied the effect of the size of the square pillars and it is found the droplet spreads less in case of smaller size of pillars for both radially increasing and decreasing surfaces.

8:11AM Q21.00003 Cassie-Baxter to Wenzel Transition and a New Phenomenon called “the Wenzel Deviation”. ARASH AZIMI, CHAE ROHRS, PING HE, Department of Mechanical Engineering, Lamar University, Beaumont, TX 77710 — In general, on rough surfaces, two wetting regimes are possible: (1) the Cassie-Baxter state, in which the droplet sits on top of rough structures, and (2) the Wenzel state, in which the droplet completely sinks into the rough structures. In this talk, we present a numerical study of the Cassie-Baxter to Wenzel transition using a series of 3D simulations for a water droplet on micro-patterned substrates, in which the pillar height and spacing are systematically varied. The contact angles for each case are measured and compared with either the Cassie-Baxter or Wenzel equation. The total surface energy and its time evolution are discussed in detail. Energy barriers for the wetting transition are addressed. Measured contact angles show an excellent agreement for the Cassie-Baxter state, while for the Wenzel state, we find a systematic deviation from the Wenzel equation when the pillar size is large. The critical pillar size, above which the Wenzel deviation is outstanding, is identified based on simulation results and thermodynamic calculations. A modified Wenzel equation is developed to account for the Wenzel deviation.

8:24AM Q21.00004 Walking, Climbing, Bursting, and Shooting: Complex Dynamics in Drops on Vibrated Substrates1. YESH KAOHUA, Imperial College London, SEUNGWON SHIN, Hongk University, South Korea, JALEL CHERGUI, DAMIR JURIC, LIMSI, CNRS, France, RICHARD CRASTER, OMAR MATAR, Imperial College London — We use direct numerical simulations (DNS) to study the phenomena observed in the work of Brunet et al. (Phys. Rev. Lett., 99, 144501, 2007). Here a drop can climb up an inclined surface when it is subjected to a vertical oscillation in the presence of a gravity. In this talk, we present a detailed study of these climbing phenomena using DNS with a generalized Navier boundary condition in the context of a front-tracking-based multiphase method. Further detailed numerical simulations in the context of vibrated droplet are extended to different vibration configurations (horizontal, vertical, and oblique) in order to explain how these climbing phenomena occur leading to regimes characterised by droplet ‘walking, ‘bursting, and ‘shooting.’

1We thank Dr. M. Costalonga and Dr. P. Brunet for helpful discussions. Funding from PETRONAS/Royal Academy of Engineering, and EPSRC (grant number EP/K003976/1) is gratefully acknowledged.

8:37AM Q21.00005 Experimental and numerical study of wetting liquids rising up on the outer surface of a nozzle in the dripping regime. ERFAN SEDIGHI, ABOLFAZL SADEGHPOUR, Department of Mechanical and Aerospace Engineering, University of California, Los Angeles, HANGJIE JI, CLAUDIA FALCON, Department of Mathematics, University of California, Los Angeles, Y. SUNGTAEK JU, Department of Mechanical and Aerospace Engineering, University of California, Los Angeles, ANDREA BEROTTOZI, Department of Mathematics, University of California, Los Angeles — Well-wetting liquids exiting small-diameter nozzles in the dripping regime rise up along the outer nozzle surfaces. This is problematic for certain fuel injectors and direct contact heat and mass exchangers that incorporate a dense array of nozzles to distribute liquids. Such flows along nozzle outer surfaces are governed by the interplay of surface tension, non-uniform pressure within a pendant drop, gravity, and viscous forces. We experimentally and numerically (by solving the full Navier-Stokes equations) study these flows for nozzle outer diameters ranging from 0.7 to 3.2 mm, mass flow rates ranging from 0.0002 to 0.035 grams/second (g/s), and liquid viscosities ranging from 4.6 to 970 mPa.s. The apparent height of a liquid meniscus on the nozzle surface was determined by analyzing video images. We also develop an approximate analytic model to capture the dynamics of meniscus rising. Our results show that, for a single nozzle, while the mass flowrate is relatively low, the rate of initial rise decreases with further decreasing the mass flow rate. On the other hand, for flow rates higher than 0.006 g/s and viscosities less than 100 mPa.s, the rate of initial rise is almost constant.

8:50AM Q21.00006 Contact angle and interface geometry immediately after rapid initiation of the contact line motion. TAKAHIRO ITO, Chubu University, KENJI KATO, TATSURO WAKIMOTO, Osaka City University — The dynamic contact angle on non-smoothed surface generally shows different variation from that expected by the theory (eg, Cox, J. Fluid Mech., 1985). One plausible mechanism for such deviation can be the local stick-slip motion of the contact line. In this study a model is developed to describe interface deformation induced by the rapid initiation of the contact line motion. The model is based on the balance of the normal stress on the surface under the Stokes approximation. The local surface stress is modified from the expression by Huh et al (J Colloid Int Sci, 1971). The contact line velocity is replaced with ‘characteristic velocity’ in order to take the lag of the time development of the local velocity relative to the contact line speed. The finite speed of the transfer of the interface deformation is also modeled. These modifications are combined with the conventional Hoffman-Tanner-Voinov relation for the dynamic contact angle and the contact line speed. The results obtained with the developed model shows good agreement with the experimental results.

9:03AM Q21.00007 ABSTRACT WITHDRAWN
ZHICHE M. YUAN, MITSUHIRO MATSUMOTO, RYOICHI KUROSE, Department of Mechanical Engineering and Science, Kyoto University; KUROSE LAB TEAM — Super-hydrophobic surfaces are reported as promising candidates for self-cleaning, anti-icing, and dropwise condensation. Therefore, there are some experimental studies and numerical simulations of droplets on hydrophobic walls. However, regarding the durability issues, an alternative technology, improving the surface wettability by grooves, has drawn much attention, whereas the wettability of droplet on groove-decorated substrate has not been fully studied. In this study, a 3-D numerical simulation employing the Coupled Level-Set and Volume of Fluid (CLSVOF) scheme, and the Continuum Surface Force (CSF) method are applied to a liquid drop on rigid substrate, and the validity is investigated by comparing with the experiment. The numerical models are extended to predict the dynamics of a droplet on groove-decorated substrate. The results show that our numerical methods perform well on tracking the move of a droplet on micro-grooved surfaces. In addition, decoration by micro-grooves could be a useful fabrication technology to improve the surface wettability and develop robustness and durability of super-hydrophobic surfaces.

1This research was partially supported by MEXT (Ministry of Education, Culture, Sports, Science, and Technology) as Priority issue on Post-K computer.

9:29AM Q21.00009 Title: Complex Wetting: Flow profiles close to three-phase contact lines1.  
BENEDIKT B. STRAUB, HENRIK SCHMIDT, FRANZISKA HENRICH, Max Planck Institute for Polymer Research, MAS-SIMILIANO ROSSI, Technical University of Denmark, CHRISTIAN J. KAHLER, Bundeswehr University Munich, HANS-JURGEN BUTT, Max Planck Institute for Polymer Research, GÜNTHER K. AUERNHAMMER, Leibniz Institute for Polymer Research — Wetting and dewetting behavior on solid surfaces is the crucial process underlying many natural phenomena as well as technical applications. We focus on the technically relevant wetting and dewetting behavior of surfactant solutions. In recent studies, focus laid on the influence of surfactants on macroscopic quantities like the contact angle. To explore the origin of the decrease of the contact angle for increasing surfactant concentration and velocity, we focus on the flow close to the contact line. Therefore, we measure three-dimensional flow profiles with an astigmatism particle tracking velocity setup. The results show that surfactants cause a deviation of the flow field in comparison to theoretical predictions for pure liquids. In the case of a receding contact line, a new air-liquid interface is formed at the three phase contact line. The surfactant concentration at the freshly formed interface is, in comparison to the already existing air-liquid interface, not in equilibrium to the bulk surfactant concentration. This causes Marangoni stresses in the direction of the contact line along the interface. This Marangoni stresses oppose the bulk flow close to the air-liquid interface and causes a deviation of the flow field.

1The authors thank the Deutsche Forschungsgemeinschaft (DFG) for supporting this work through the SFB1194 (project A06).

Tuesday, November 26, 2019 7:45AM - 9:42AM —  
Session Q22 Drops: Instability and Breakup I —  
604 - Larry Li, HKUST

7:45AM Q22.00001 ABSTRACT WITHDRAWN —  

7:58AM Q22.00002 Direct Numerical Simulations of the Three Dimensional Dynamics of Surfactant Laden Retracting Ligaments1.  
RICARDO CONSTATE-AMORES, ASSEN BATCHVAROV, LYES KHAOUADJI, Imperial College London; SEUNGWÖN SHIN, Hongik University, South Korea; JALEL CHERGUI, DAMIR JURIC, LIMSI, CNRS, France; RICHARD CRASTER, OMAR MATAR, Imperial College London — We present three-dimensional direct numerical simulations for the retraction of surfactant-laden Newtonian ligaments. In the absence of surfactant, the dynamics is described non-dimensionally by the ligament aspect ratio and the Ohnesorge number, which relates the viscous forces to surface tension forces. The addition of surfactant leads to surface tension gradients resulting in Marangoni stresses on the interface. We consider surfactants, which are either insoluble, where molecules remain only on the interface, or soluble which allows surfactant mass transfer between the bulk and the interface via adsorption and desorption mechanisms. In addition to the ligament aspect ratio and Ohnesorge number, we take into account variation of the Peclet, Biot, and elasticity numbers to elucidate the physical mechanisms which control the dynamics of the ligament in the presence of surfactant. An analysis of the effect of surfactant concentration, surface tension, tangential velocity, and Marangoni stresses is provided and of their role in partial/complete ligament breakup suppression.

1Funding from BP, EPSRC, PETRONAS, and Royal Academy of Engineering.

8:11AM Q22.00003 Dynamics of Viscoelastic Filaments.  
SUMEET THETE, Air Products and Chemicals, Inc.; BRADLEY WAGONER, School of Chemical Engineering, Purdue University; PRITISH KAMAT, Dow Inc.; MICHAEL HARRIS, OSMAN BASARAN, School of Chemical Engineering, Purdue University — In processes as diverse as ink-jet printing and crop spraying, slender liquid filaments are formed. These filaments may contract into a single drop or breakup into multiple smaller drops. In the aforementioned and most other applications, the latter outcome is highly undesirable as it will mar the printing quality or increase the portion of the spray fluid that can drift. In many applications, the working fluids may contain additives that render them viscoelastic in nature. While the dynamics of Newtonian filaments have been studied extensively, that of viscoelastic filaments remains largely unexplored. Using the conformation tensor formalism of Pasquali and Scriven (Pasquali and Scriven, J. Non-Newtonian Fluid Mech., 2004) implemented in a numerical algorithm based on the SUPG/FEM formation, we present the results of a study on the fate of viscoelastic filaments.
The shape of a recoiling liquid filament

8:37AM Q22.00005 Effect of ICs on dynamics of contracting filaments

8:50AM Q22.00006 Application of Digital Inline Holography to Quantify the Influence of Dilute Oil-In-Water Emulsions on Sprays

9:03AM Q22.00007 Shock-induced atomization of water droplets

9:16AM Q22.00008 Simultaneous LIF/shadowgraphy visualization of droplet breakup in high-speed flows

9:29AM Q22.00009 Viscoelastic Sphere Impact onto a Water Pool
Turbulent drag reduction using biopolymers and bio-inspired superhydrophobic surfaces

Skin friction accounts for over 50% of the total drag on ships, and its effective mitigation can yield significant savings in fuel, operating costs, and emissions. Despite considerable promise as an effective drag reduction strategy, high molar mass additives have been largely precluded from commercial use due to the high cost of synthetic polymers. In this context, we investigate the aqueous mucilage extracted from seeds such as flax, chia and psyllium as viable, cost-effective and eco-friendly alternatives to synthetic water-soluble polymers. Using frictional drag measurements performed in a bespoke Taylor-Couette apparatus, we show that aqueous mucilage displays comparable drag reduction efficacy as polyethylene oxide, a common synthetic polymer, at a much lower cost. We study the effects of salinity and shear-induced chain scission, and explore the use of cross-linking agents to augment the drag reduction performance of the dissolved chains. Finally, we investigate the use of scalable, randomly rough superhydrophobic walls (inspired by the natural texture on the leaves of the lotus and other plants) as a passive means to mitigate turbulent skin friction, and its synergistic use in conjunction with dilute polymer solutions to enhance the overall reduction in frictional drag.

We appreciate funding from NSF award 1705958

Lubricated tribology of soft patterned substrates

We study the changes in the total frictional drag as a function of riblet wavelength and amplitude (as the key geometric parameters) and mainly centered around riblets with V-groove profiles. Here, we will discuss the impact of the variations in the cross-sectional profiles of the riblets on the changes in the total frictional drag. Using a custom-designed Taylor-Couette cell with 3D-printed texture-covered rotors, we explore the impact of different riblet textures and discuss the effect of concave vs. convex cross-sectional shapes on the ability of the riblet surfaces to reduce the total frictional torque exerted on the textured rotors. Lastly, we will compare the effectiveness of riblets with curved-profiles against the conventional V-grooves (linear profiles).

Impact of Curved Profiles on the Drag-Reducing Ability of Riblet-Textured Surfaces

We study the changes in the total frictional drag as a function of riblet wavelength and amplitude (as the key geometric parameters) and mainly centered around riblets with V-groove profiles. Here, we will discuss the impact of the variations in the cross-sectional profiles of the riblets on the changes in the total frictional drag. Using a custom-designed Taylor-Couette cell with 3D-printed texture-covered rotors, we explore the impact of different riblet textures and discuss the effect of concave vs. convex cross-sectional shapes on the ability of the riblet surfaces to reduce the total frictional torque exerted on the textured rotors. Lastly, we will compare the effectiveness of riblets with curved-profiles against the conventional V-grooves (linear profiles).

Efficient and sustainable drag reduction surface of marine algae Miyeok (Undaria pinnatifida)

Miyeok has a slippery surface in which mucilage works as a lubricant. In this study, we investigated the drag reduction effect of the mucilage-covered surface of Miyeok using a scanning white-light interference technique to evaluate the slip length. Pressure drop and skin friction effects were measured to estimate the slip effect and the corresponding drag reduction rate. The Miyeok surfaces tested in this study were found to have a slip length of 101 μm and the pressure drop was 26% reduced due to the morphological characteristics and slippery mucilage. With these effects, the mucilage surface exhibits skin friction reduction up to 35%. The present results would be helpful for understanding the drag reduction mechanism of Miyeok and developing a bioinspired LIS.

Fricion Reduction Effects of Wetted Microtexturing in Microchannel Flow

Microchannel flows are widely used in applications where small diffusion length scales are important, such as in microscale heat exchangers for electronic cooling. However, these small length scales also translate into high pumping power requirements. One possible way to alleviate the large viscous pressure losses associated with this inherent dimensional constraint is to introduce side trenches in a micro-channel to help lower the skin drag. The flow over these transverse trenches may experience two wetting states: Cassie-Baxter and Wenzel. In both states the trapped air or water can act like a cushion resulting in less shear stress. However, it has been shown that sometimes the air-water interface in the Cassie-Baxter state might act like a solid boundary due to contamination. Concurrently, penetration of the flow inside the trenches can induce the pressure drag alongside the skin drag. Therefore, the Wenzel state in the trenches can lead to a trade-off between skin and pressure drag. The aim of this work is to understand the geometrical effect that different micro-textures have on the total drag reduction by testing trenches with different aspect ratios and measuring the pressure drop through the micro-channel.

We appreciate funding from NSF award 1705958
1. Author for correspondence

9:03 AM Q23.00007 Quantification of Laminar Drag Reduction on Liquid-Infused Structured Non-Wetting Surfaces, SANDEEP HATTE, KARTHIK NITHYANANDAM, RANGA PITCHUMANI, Department of Mechanical Engineering, Virginia Polytechnic Institute and State University — Liquid-infused structured non-wetting surfaces offer alternating no-slip and finite slip boundary conditions to the fluid flow, resulting in an effective non-zero finite slip at the interface. As a result, liquid-infused structured non-wetting surfaces offer reduced friction (drag reduction) at the interface in comparison to a bare smooth surface offering no-slip boundary condition throughout. In the present work, an analytical model is developed to quantify the effective slip length, drag reduction and friction coefficient on liquid-infused structured non-wetting surfaces under laminar fluid flow conditions. The model takes into consideration a typical structured non-wetting surface as a superposition of periodically patterned longitudinal and transverse striped geometries. The analytical model covers a full range of structural anisotropy and homogeneity and is valid for an entire range of partial slip length in the infused liquid region. Effective non-zero slip length and drag reduction data predicted from the present model show a good agreement when compared with a number of experimental and computational studies from the literature.

9:16 AM Q23.00008 Bound on the drag coefficient for flow past a flat plate using the background method, ANUJ KUMAR, Department of Applied Mathematics, Baskin School of Engineering, University of California, Santa Cruz — The background method has been a successful tool in finding bounds on mean quantities, such as heat and mass transfer, drag force, and others. Until now, most of the applications of this method focused on flows confined between planar boundaries, such as Rayleigh-Bénard convection, Poiseuille flow, and Couette flow. The extension of this method to unconfined flows, such as flow past a sphere, has remained elusive due to a number of mathematical difficulties. In particular, proving bounds on the drag coefficient for flow past an object is an open problem. We will demonstrate that the case of flow past a flat plate avoids some of these difficulties, enabling us to apply the background method to this problem for the first time. We show that at high Reynolds number, the drag coefficient is bounded by a constant. We compare our finding with observations. Finally, we make a few remarks about the issues in using the background method for flow confined between rough boundaries or flow past objects of non-zero volume.

9:29 AM Q23.00009 Numerical Study of Ice Accretion over Aircraft Wings Using Delayed Detached Eddy Simulation, SIBO LI, Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago IL, USA, ROBERTO PAOLI, Leadership Computing Facility, Argonne National Laboratory, Lemont IL, USA — Ice accretion on aircraft surfaces has been the principal cause of several flight accidents in the past and represents now a source of major concern in aviation. It is a complex Multiphysics phenomenon that includes fluid dynamics, heat transfer and multi-phase flows. In this study, a mathematical model based on the delayed detached eddy simulation (DDES) is developed to study the ice accretion on 3D aircraft wings. The model is validated by comparing the computed results with experimental data. For the air flow field, the statistical results, instantaneous flow fields, and pressure fluctuations are first analyzed. Then, the droplet-phase governing equations are solved to obtain the droplet collection efficiency. To best mimic the droplets impingement, a permeable wall boundary condition is proposed. Then, the thermodynamic process of ice accretion is built based on the classical Messenger model but further including the freezing fraction as a changing variable in the icing simulation. Thus, the ice amount generated at each time step can be obtained. The DDES is able to capture the turbulent flowfield around the iced wing, which makes this model useful for not only predicting ice accretion and studying the effect of ice shape on the air flow field as well.

Tuesday, November 26, 2019 7:45 AM - 9:42 AM — Session Q24 Drops: Complex Fluids

7:45 AM Q24.00001 Numerical Simulation of Interfacial Flows of a Hydrogel, LEI LI, Department of Chemical and Biological Engineering, University of British Columbia, Vancouver V6T 1Z3, Canada., PENGTAO YUE, Department of Mathematics, Virginia Tech, Blacksburg, VA 24061-0123, The United States of America. — Hydrogels are crosslinked polymer networks swollen with an aqueous solvent, and play central roles in biomicrofluidic devices. In such applications, the gel is often in contact with a flowing fluid, thus setting up a fluid-hydrogel two-phase system. Using a recently proposed model (Y.-N. Young et al, Phys. Rev. Fluids 4, 063601, 2019), we treat the hydrogel as a poroelastic material consisting of a neo-Hookean polymer network and a Newtonian viscous solvent, and numerically study the motion and deformation of gel drops suspended in viscous flows. The gel-fluid interface is tracked by using the Arbitrary Lagrangian-Eulerian method that maps the interface to a reference configuration. The interfacial deformation is coupled with the fluid and elasticity governing equations into a monolithic solution algorithm using the finite-element library deal.ii. Our numerical simulation of a hydrogel drop in sedimentation and shear flow shows that it deforms in ways that differ from that of a viscous drop or elastic particle, and the solvent perfusion can have a significant effect on the hydrogel dynamics.

7:58 AM Q24.00002 Stability of chiral colloidal droplets, LEROY JIA, Flatiron Institute, EPHRAIM BILILIGN, University of Chicago, MICHAEL SHÉLLEY, Flatiron Institute & NYU, WILLIAM IRVINE, University of Chicago — We report experiments on a two-dimensional cohesive chiral fluid consisting of millions of spinning colloidal magnets suspended in water. Droplets made of this fluid are observed to swirl around and collapse into a single central superdroplet in a process reminiscent of the accretion of matter during the formation of a black hole. We put forth a minimal but complete hydrodynamic description of this active chiral fluid and use it to analyze the stability of the droplet states.

1 National Science Foundation MRSEC Program at The University of Chicago (Grant DMR-1420709) and a Packard Fellowship
8:11AM Q24.00003 Central spot shaped deposition from colloidal droplet evaporation under enhanced Marangoni effect, FEI DUAN, JUNHENG REN, Nanyang Technological University — Evaporation of a colloidal droplet on a substrate can result in a residual deposit near the three-phase line, driven by the capillary flow, and form coffee ring. Under a higher evaporation rate, the more nonuniform temperature can be generated at liquid-gas interface of the sessile droplet. The evaporation induced Marangoni flow can be observed, the particles can be carried to move toward the centerline and aggregate at the center area of the droplet. We have developed a three-dimensional (3D) diffusion limited aggregation Monte Carlo method to simulate the dried patterns under the enhanced Marangoni effect. The particle motion is controlled by calculating the probabilities of six moving directions. The interactions amongst particle to particle, particle to liquid, particle to substrate have been taken into account. The results show that the final dried residuals can be formed to with a central spot inside a thin coffee ring by enhancing the Marangoni flow during drying a colloidal droplet. The results are comparable to the deposited patterns under the reduced pressure evaporation conditions. The drying patterns have been analyzed as well.

8:24AM Q24.00004 The role of surfactant in evaporation and deposition of bi-solvent biopolymer droplets, DONG-OOK KIM, ARIF ROKONI, CHUNXIAO CUI, LI-HSIN HAN, YING SUN, Drexel University — The quality of bioprinting is determined by the solvent evaporation and deposition processes of biopolymer droplets, during which instantaneous viscosity and surface tension changes occur. Such dynamics is complex and not well understood. Using high-speed interferometry and particle image velocimetry, we directly observe in real time the instantaneous drop shape and micro flows inside inkjet-printed evaporating gelatin drops containing glycerol and water. It is observed that, for bi-solvent gelatin drops with surfactants, highly viscous gelatin and glycerol accumulated near the pinned contact line at an early stage suppress the evaporation-driven outward flow and create a stagnation zone near the contact line. Lower surface tension at the contact line as compared to the drop apex induces a strong Marangoni recirculation, which in conjunction with a stagnation zone in the contact line region, causing the drop shape to transition from a spherical cap to a volcano shape during evaporation. In contrast, without surfactants, the Marangoni-induced Marangoni flow can be observed, the particles can be carried to move toward the centerline and aggregate at the center area of the colloidal droplet. We have developed a three-dimensional (3D) diffusion limited aggregation Monte Carlo method to simulate the dried patterns under the enhanced Marangoni effect. The role of surfactant in polymer droplet deposition with water-only solvent is also investigated and compared against that of bi-solvent drops.

8:37AM Q24.00005 Evaporation-driven Fracture of Colloidal Drops Undergoing a Sol-gel Transition1, ARANDEEP UPPAL, Imperial College London, MATTHEW HENNESSY, University of Oxford, RICHARD CRASTER, OMAR MATAR, Imperial College London — Evaporation of liquid from a colloidal suspension can lead to a sol-gel transition whereby the mixture is transformed into a soft, gel-like material consisting of closely-packed particles with liquid-filled voids. If a drop of colloidal fluid is placed on a solid substrate, then non-uniformities in the evaporation rate lead to a gelation front that propagates from the contact line towards the center of the drop. Afterwards, uniformly spaced, radially aligned cracks begin to form near the contact line which themselves propagate into the bulk. By varying the composition of the drop, a myriad of striking fracture patterns can be observed. To describe this evaporation-driven fracture process, we have developed a fully-coupled poroelastic-damage model. The model is systematically reduced by exploiting the small contact angle of the drop and the resulting equations are solved using the finite element method. The model confirms that fracture is driven by the generation of tensile stresses at the contact line caused by the adhesion of the solid matrix to the substrate and volumetric contraction due to fluid loss. Our simulations are able reproduce the experimentally observed fracture patterns and reveal that slip plays a key role in selecting the resulting morphology.

1Engineering and Physical Sciences Research Council, UK, Centre for Doctoral Training in Fluid Dynamics across Scales (EP/L016230/1)

8:50AM Q24.00006 To jam or not to jam?, RISHTI ARORA, MICHELLE DRISCOLL, Northwestern University — The extraordinary hydrodynamic phenomenon of drop impact has inspired numerous studies exploring a variety of Newtonian and complex fluids. Although this is a century-old problem, there are only a few studies on the impact dynamics of colloidal suspension drops. The impact of a suspension drop provides a unique model system to probe the complicated interplay between hydrodynamic instability and the non-trivial rheology of complex fluids. Here, we present a comprehensive study of colloidal suspension drop impact to explore the rich flow behaviors in conditions that are inaccessible via conventional rheometry. We study the dynamics of suspension drops by impacting a millimetric size droplet on a solid glass surface while varying the volume fraction and impact conditions. We find that the extent of spreading decreases with increasing volume fraction or decreasing impact velocity. Moreover, we present a state diagram that delineates the inertial spreading regime from thickening regime and the jammed solid regime. Furthermore, we observe a variety of elastic behaviors which appears in the jammed regime and are controlled by adjusting volume fraction and impact velocity.

9:03AM Q24.00007 Onset of rebound suppression in non–Newtonian droplets post–impact on superhydrophobic surfaces, DEVRAJAN SAMANTA, SOUMYA RANJAN MISHRA, PURBARUN DHAR, Indian Institute of Technology Ropar — Droplet deposition after impact on superhydrophobic surfaces has been an important area of study in recent years for its potential application in reduction of pesticides usage. Minute amounts of long chain polymers added to water has been known to arrest the droplet rebound effect on superhydrophobic surfaces. Previous studies have attributed different reasons like extensional viscosity, dominance of elastic stresses or slowing down of contact line in retraction phase due to stretching of polymer chains. The present study attempts to unravel the existence of critical criteria of polymer concentration and impact velocity in the inhibition of droplet rebound. The impact velocity will indirectly influence the shear rate during the retraction phase, and the polymer concentration dictates the relaxation timescale of the elastic fluids. Finally, we show that the governing Weissenberg number (at onset of retraction), which quantifies both the elastic effects of polymer chains and the hydrodynamics, is the critical parameter in determining the regime of onset of rebound suppression, and that there is a critical value which determines the onset of bounce arrest. The previous three causes, which are manifestations of elastic effects in non-Newtonian fluids, can be related to the proposed Weissenberg number criterion.

9:16AM Q24.00008 Drop impact of extensible yield-stress fluids, SAMYA SEN, RANDY H EWOLDT, University of Illinois at Urbana-Champaign — We study the role of extensional rheology on the impact behavior of drops of elastoviscoplastic fluids on thin films, showing that a recently proposed and successful dimensionless group for viscoplastic fluids fails to predict impact regimes when elastic extensional effects are significant. We do this by creating a new formulation for an extensible yield-stress fluid in which extensional properties vary dramatically. The non-Newtonian fluid is an aqueous suspension of Carbopol microgel particles (a well-studied system) with the addition of high molecular weight poly(ethylene oxide) (PEO) at varying concentration. The fluids fit into the new paradigm of extensible yield-stress fluids [1]. Drop impacts onto substrates coated with the same material are recorded using high-speed cameras, and different impact regimes are identified as a function of droplet size, velocity, coating thickness, and droplet rheology. Whereas the previous dimensionless group only involves steady shear rheological properties (yield stress and Bingham plastic viscosity), we consider additional modification to account for the elastic extensional effects. [1] Nelson, A.Z., R.E. Bras, J. Liu, and R.H. Ewoldt, “Extending yield-stress fluid paradigms”, J. Rheol. 62, 357 (2018)

1 Funded by National Science Foundation, CAREER Award, CBET-1351342
9:29AM Q24.00009 Drop impact of thixotropic yield-stress fluids1, RANDY H EWOLDT, SAMYA SEN, ANTHONY G MORALLES, University of Illinois at Urbana-Champaign — We use high-speed imaging to study the effect of thixotropy on drop impact of thixotropic- viscoplastic fluids onto thin films. Using a dimensionless group proposed earlier for predicting impact behavior of Carbopol, a glassy aqueous suspension of soft microgel particles, we predict the behavior of an aqueous suspension of Laponite, a colloidal clay with an attractive gel microstructure. For rejuvenated samples of both materials, the dimensionless group separates various impact regimes, gives a constant critical value for stick-slip transition as a function of dimensionless coating thickness, and gives a similar critical value across the varying microstructures studied. This is remarkable considering the different chemistries and microstructures involved. The group is less effective at predicting impact behavior of thixotropically aged Laponite suspensions, demonstrating the role of thixotropy in drop impact with viscoplastic fluids. We tune this dimensionless group to include thixotropic effects by proposing rheological techniques to estimate shear properties of thixotropically aged samples. The modified group is tested with a wide range of experimental conditions and is shown to be effective in predicting the drop impact regimes of aged Laponite samples.

1Funded by National Science Foundation, CAREER Award, CBET-1351342

Tuesday, November 26, 2019 7:45AM - 9:42AM — Session Q25 Drops: General II 607 - Anne Juel, University of Manchester

7:45AM Q25.00001 Experiments in oil-water ring-sheared drop1, SHANNON GRIFFIN, PATRICK McMACKIN, FRANK RILEY, SHREYASH GULATI, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — Launched to the International Space Station (ISS) in late July 2019, the ring-sheared drop (RSD) is a containerless reactor where surface tension provides fluid containment, and shear is conveyed primarily through surface shear viscosity. The RSD is a 2.5 cm diameter drop constrained by two thin rings. A stationary ring contacts the drop in one hemisphere, and a rotating ring contacts in the opposite hemisphere. The RSD allows investigation of how thin-film flow can modulate the role of Marangoni convection. Amyloid fibrils of certain brain proteins play central roles in some neurodegenerative diseases, such as Alzheimer’s or Parkinson’s. The RSD was studied in the lab using a density-matched silicone oil-water system. Through these experiments, previous computational work on the RSD was verified. Drop deformation was found to be a balance between viscous, inertial, and capillary forces. These laboratory experiments provided some evidence for the robustness of the RSD.

1This work was supported by NASA grant NNX13AQ22G.

7:58AM Q25.00002 Formation and evolution of thin water films encapsulating oil droplets crossing an oil-water interface. JOSEPH KATZ, OMRI RAM, Johns Hopkins University — This study examines phenomena occurring as mm-scale oil droplets rise through water, cross an oil-water interface, and then coalesce with a layer of the same oil. Inline holography and planar laser induced fluorescence are used for examining interfaces involving layers of silicone oil of different viscosity and hexadecane above both purified and nearly refractive index-matched sugar water. They show that for all cases, after crossing the interface, thin water films remain around these droplets, preventing them from mixing with the bulk oil. Subsequently, in a slow process, film segments located close to the contact point with the interface are attracted to the bulk water, presumably by electrostatic forces, causing the droplet to flatten, and creating a kink, where the film begins to break up into submicron droplets. The region of submicron droplets propagates from the peripheral kinks inward to the center of the droplet, and they begin to diffuse. These processes occur in less than one minute for a 4 cst oil, and about one hour for a 50 cst oil, preventing the merging of the droplet with the surrounding oil for much longer than previously presumed. These phenomena do not occur when water droplet descends across the oil-water interface, namely oil films do not form.

8:11AM Q25.00003 Nonlinear Phenomena in Trajectories of Spherical Drops with Incompressible Surfactant in Combined Buoyant and Marangoni-induced Motion. MICHAEL ROTHER, University of Minnesota Duluth — We consider the interactions of two sedimenting, spherical drops in a temperature gradient with negligible thermal convection. The drops are covered with a nearly uniform surfactant film. When drop and surrounding fluid inertia lose symmetry, as indicated by small Stokes and Reynolds numbers, respectively, the governing equations are linear, and the trajectories are symmetric. In the linear case, it is possible for two drops to fall as a pair with constant horizontal separation, but the drops must have the tandem arrangement initially. Moreover, a saddle point can occur in the trajectory phase plane. When drop inertia is considered, as measured by a finite Stokes number, with surrounding fluid inertia still unimportant, the governing equations become non-linear, and the trajectories lose symmetry. Asymmetric limit cycles can be observed. In addition, trajectories with a constant, finite horizontal gap become stable, with retrograde motion also possible. For small water drops in air, at a fixed drop size ratio, the collision efficiency vanishes, depending on the relative strength and orientation of the gravitational and thermocapillary driving forces.

8:24AM Q25.00004 ABSTRACT WITHDRAWN —

8:37AM Q25.00005 Role of Laplace pressure in the equilibrium of a pendent drop highlighted by a particle loading, L. LECACHEUX, A. SADOUFI, D. CASSAN, A. DURI, UMR IATE 1208 CIRAD/INRA/SupAgro/Universite Montpellier,2 Place Viala,34060 Montpellier,France, T. RUIZ, UMR QualiSud 95 CIRAD/Universite Montpellier,15 av Charles Flahault,34093, Montpellier — A pendent drop at the end of a capillary needle remains in near equilibrium just before breaking: the action of the capillary force must oppose the weight of the drop, to which should be added the force due to Laplace pressure (LP). Thus, for a given fluid and a fixed wet perimeter, its maximum mass should be constant. We modulated the LP by modifying the main curvatures of the drop and we observed the impact on its mass. Drops were loaded with glass beads of increasing mass and induce a stretching of the drops that modulates their main curvatures. Volumes and curvature radii are measured by image analysis. These loading experiments highlight the increase of LP with the loading and the non-linear decrease of the drop mass. However, we observe that the liquid mass in the loaded drop decreases linearly with the increase of the bead mass without verifying the mass balance. Such a result is included in a master curve which highlights the role of the LP in the equilibrium of a pendent drop just before its rupture. It challenges the validity of the well-known Tate’s law and allows the setting of functional ranges for capillary micromanipulators.
films in a steel plate illuminated by a high energy laser1. MARY LANZEROTTI, United States Military Academy, K. BRAKK, Susquehanna University, K. ALLEN, J. HARTKE, United States Military Academy, A. HIRSA, Rensselaer Polytechnic University — This talk presents preliminary observations of the formation of a single liquid drop of molten steel following bursting of a thick liquid film formed by illumination of a thin vertical steel plate by a 1075-nm continuous-wave 1000W Ytterbium fiber laser. Images of the initial hole captured by a high-speed digital camera at room temperature conditions show a rapidly expanding hole. Liquid then gathers at the lower part of the hole, forming a liquid lump where the pre-burst bulge was located. The liquid lump vibrates twice as it settles into a sphere. Gravity pulls the liquid drop down, to form a neck between the rim of the hoop and the drop. The neck pinches off, detaching a drop. The neck retracts slightly before it freezes with a small droplet that appears to form at the bottom of the neck before the neck freezes. The falling drop oscillates as it falls until it hits the mount surrounding the steel plate. The high-speed images also show an ejected droplet that is launched and then falls, apparently in front of the plate. No satellite drops are observed.

1Army JTO, ARO

9:03AM Q25.00007 Onset time of fog collection using a single wire. YOUSHUA JIANG, CHRISTIAN MACHADO, SHAAN SAVARIYAN, NEELESH PATANKAR, KYOO-CHUL PARK, Northwestern University — Fog collection is a promising solution to water scarcity while also being of vital importance in industrial processes. To date, many studies have investigated the fog collection rate, a parameter that denotes the average performance over a period of time. However, the initial period (referred to as onset time) between the start of the fog flow and the collection of the captured liquid (a delay in time caused by droplet growth to a critical weight) has not been understood. A longer onset time results in a more serious clogging issue that deteriorates the collection rate and, therefore, understanding this phenomenon is important. Here, we study how the onset time is determined by the capture and transport of fog droplets using single, vertical wires with various surface wettabilities and diameters, under different wind speeds. We derive a scaling law that correlates the onset time with the fog capture process and droplet-surface retention force, governed by aerodynamics and interfacial phenomena, respectively. In particular, the onset time decreases with an increasing rate of fog capture or a decreasing droplet-surface retention. This study introduces a new aspect in the evaluation of fog collection and provides insights for the design of fog collectors.

9:16AM Q25.00008 Clogging-enhanced fog harvesting on the hole pattern1. JIHYE PARK, CHANGJE LEE, HYESUNG CHO, MYOUNG-WOON MOON, SANGYOUNG LEE, SEONG JIN KIM, Korea Institute of Science and Technology — Clogging has been considered as a drawback in fog-harvesting as it hinders local airflow around a hole. However, in this study, it is demonstrated that the clogging effect on the fog-harvesting performance differs between the stable clogging and the unstable clogging. Unlike the stable clogging that is stably stuck in the hole, the unstable clogging is observed to form and break up repeatedly during the fog-harvesting process. Due to this break-up, fog flow through the unstably-clogged mesh experiences much less pressure drop than that across the stably-clogged mesh even though those meshes have the same shade coefficient. Moreover, it is found that the clogging can be used even for enhancing the fog-capturing efficiency than the clogging-free mesh with the same shade coefficient by increasing the effective shade coefficient to collide with more fog drops, which suggests that the clogging could be not a simple drawback. Rather, especially for the unstable clogging, it can be used to optimize the fog-harvesting performance with maintaining low pressure drop. Moreover, the particle image velocimetry is performed to quantify the increase in the effective shade coefficient of the clogged mesh by measuring the turbulent kinetic energy of the fog flow to the mesh. 1This research was a part of the project titled ‘Development of clean-up technology for spilled oil and floating HNS using nanostructures’, funded by Korea Coast Guard, Korea.

9:29AM Q25.00009 Droplets sliding on fiber arrays. FLORIANE WEYER, University of Liege, ALEXIS DUCHESNE, University of Lille, NICOLAS VANDEVALLE, University of Liege — At a time when water becomes a scarce commodity in the most arid regions, fog harvesting has emerged as a sustainable alternative way to supply drinkable water. The small droplets from the fog impact the mesh net and, as more and more water hits the mesh fibers, the droplets fuse together and slide along the fibers due to gravity. Even though the collection process has been widely studied, the motion of the droplets on the fibers remains an open question. Here, we focus on the motion of droplets on an entanglement of fibers. We use a model system made of silicone oil droplets, to insure total wetting, sliding on nylon fibers. The movement of the droplet depends mostly on the volume of the drop, the fiber inclination and the fiber diameter. The results allow us to better understand the path chosen by the droplets once collected. These results could be used to improve the collection rate of the fog collection devices.

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q26 Flow Control: Turbulence 608 - Wei Zhang, Cleveland State University

7:45AM Q26.00001 Turbulent Wake Induced by a Seal-Whisker-Inspired Power Turbine Blade, WEI ZHANG, ROBERT AHLMAN, CURTIS FLACK, Cleveland State University, VIKRAM SHYAM, NASA Glenn Research Center — Power turbines operate over a large range of flow incidence and at relatively low Reynolds numbers (Re). For a fixed-wing aircraft at cruising altitude, it is challenging for current power turbine blades to maintain desirable aerodynamic performance as the Re number has dropped to substantially lower than that at sea level. Therefore, it is imperative to improve the aerodynamics of turbine blades in low Re regimes. However, the performance of state-of-the-art turbine blades for aero-propulsive systems has plateaued. Inspired by the exceptional hydrodynamics of harbor seal whiskers, this study applies the key features of the three-dimensional undulating morphology of seal whiskers to the turbine blade leading edge. Turbulent wake flows generated by a seal-whisker-inspired variable speed power turbine (VSPT) blade are quantified and compared against a baseline untreated VSPT blade in a water tunnel using Particle Image Velocimetry (PIV). Focus is on the wake velocities and turbulence statistics at a range of angles of attack (AOA = 10 to 10 degrees). Results of the seal-whisker-inspired blade will be used not only to improve the design of power turbine blades, but also to inform a wide variety of bio-inspired aerodynamic applications.
8:11 AM Q26.00003 Wall-Jet Turbulence and Mixing Control by Way of a Pulled Inlet Velocity. CRISTALE GARNICA, BERTRAND ROLLIN, Embry-Riddle Aeronautical University — Attaining skin friction reduction and increasing flow mixing is of utmost importance to enable breakthroughs in fuel efficiency, heat transfer, as well as drag and noise reduction. The turbulent plane wall-jet constitutes a typical flow configuration where turbulence phenomena associated to these engineering applications occur. Direct Numerical Simulations (DNS) are performed to investigate how skin friction and flow mixing are affected by introducing controlled perturbations, jet inlet pulsing, at the shear layer origin. The jet inlet pulse frequencies are varied in time and space to investigate its influence in the downstream domain. The forcing affects the energy of the large-scales which will consequently affect the small-scales allowing turbulence modulation. The interaction between outer layer of jet and structure near the wall, that is, near-wall jets, should be investigated. To this end, experiments of experimental and computational data from previous studies, with and without external perturbation or co-flow, are used to quantify the effect of jet inlet pulsing on wall-jet characteristics, which include eddy production, velocity profiles, maximum velocity decay, half-width jet growth, and coherent structures.

8:24 AM Q26.00004 A 2-dimensional–3-component model of turbulent flow over riblets. DAVIDE MODESTI, SEBASTIAN ENDRIKAT, Department of Mechanical Engineering, University of Melbourne, Victoria 3010, Australia, RICARDO GARCIA-MAYORAL, Department of Engineering, University of Cambridge, Cambridge CB2 1PZ, UK, NICHOLAS HUTCHINS, DANIEL CHUNG, Department of Mechanical Engineering, University of Melbourne, Victoria 3010, Australia — Riblets are streamwise-aligned grooves that are designed to reduce drag by modifying the near-wall flow with respect to that of the smooth wall. Nevertheless, drag reduction breaks down when the viscous-scaled square root of the groove area $\ell_g^+$ > 11, and this breakdown has been attributed to the formation of time-averaged secondary flow structures over riblets. This is achieved by using the 2-dimensional–3-component (2D–3C) model of Gayme et al. (J. Fluid Mech., vol. 665, 2010, pp. 99–119), in which a sustained turbulent flow is obtained by modelling the incoherent turbulent fluctuations as random forcing. We conduct 2D–3C simulations of flow over several riblet geometries and sizes and compare the results with minimal direct numerical simulations. The 2D–3C model captures the onset and the topology of the secondary flows, suggesting that they are generated by a preferential distribution of near-wall turbulence pinned by the riblet grooves. The model can be used to predict the slip velocity at the riblet crest, providing a better estimate than Stokes (purely viscous) calculations for riblets of moderate sizes, $\ell_g^+ < 20$.

8:37 AM Q26.00005 One-shot methods for nonlinear optimization of turbulent flows with heat transfer. SALEH NABI, Mitsubishi Electric Research Labs, PIYUSH GROVER, Mechanical and Materials Engineering, University of Nebraska-Lincoln, C. P. CAULFIELD, BPI & DAMTP, University of Cambridge — We consider the optimization of buoyancy-driven flows governed by Boussinesq equations using i) Direct-Adjoint-Looping (DAL), and ii) one-shot methods. Various optimization scenarios are considered: first we solve a series of inverse-design problems for which the global optimal solution is known. We demonstrate that each optimization method is able to retrieve the optimal solution in a fully turbulent regime. Next, we consider the problem of maintaining a desired temperature field with specified input energy budget. The role of an approximate Hessian as a preconditioner as well as tuned step-size for the one-shot method iterations are highlighted. It is shown, by employing an efficient optimization algorithm, the one-shot method can solve the PDE-constrained optimization problem with a cost comparable (about fourfold) to that of the simulation problem alone, and substantially cheaper than using DAL, which requires $O(10)$ direct-adjoint loops to converge. The optimization results arising from the one-shot method can be used for optimal sensor/actuator placement tasks, or to provide a reference trajectory to be used for online feedback control applications.

8:50 AM Q26.00006 Targetted modal turbulent flow control via localized heating. MATTHEW YAO, DUOSI FAN, KHALED YOUNES, JOSEPH MOULALLEM, JEAN-PIERRE HICKEY, University of Waterloo — Bidimensional empirical mode decomposition (BEMD) is an empirical method to decompose fluctuating signals into various intrinsic mode functions (IMFs); these represent different scales of the turbulent fluctuations. The scale separation flow permits an analysis of their respective contributions towards the overall skin friction of the turbulent boundary layer. We quantify the effects of selective, localized wall heating on the formation and dynamics of turbulence structures at various scales, and consequently, the effect on the skin friction for turbulent flow control. The decomposition is applied to an unheated channel flow and is compared to a channel flow with streamwise aligned heated strips. The strip spacing is dependent on the length scale of the targeted turbulent structures. The individual contribution of the various eddy sizes to the overall skin friction is then calculated and compared to the unheated base case.

9:03 AM Q26.00007 Turbulence Control in Pipe Flow by Means of Unsteady Driving. DAVIDE SCARSELLI, JOSÉ M. LOPEZ, ATUL VARSHNEY, BJÖRN HOF, IST Austria — Turbulent flows are responsible for huge energy losses in many diverse pumping applications ranging from heat exchange circuits to hydroelectric power plants. Several techniques to reduce frictional drag have been proposed over the last decades, however, very few have been tested experimentally and even less actually implemented. Based on the friction reducing properties observed in accelerating flows, we here propose a new approach to reducing drag by means of a pulsatile flow rate. We find 27% drag reduction in fully turbulent pipe flow in experiments and this is confirmed in direct numerical simulations. The optimal Reynolds number modulation is discussed. Different from many other drag reduction techniques, this operation mode does not require feedback loops, fluid additives or any modification to an existing pipeline.
9:16 AM Q26.00008 Reynolds number effect on drag control via spanwise wall oscillation in turbulent channel flows, XI CHEN, Beihang University, JIE YAO, FAZLE HUSSAIN, Texas Tech. The effect of Reynolds number ($Re$) on drag reduction (DR) by spanwise wall oscillation is studied through direct numerical simulation of incompressible turbulent channel flows with $Re$ ranging from 200 to 2000. For the non-dimensional oscillation period $T^+=100$ with maximum velocity amplitude $A^+=12$, DR decreases from 35.3% at $Re=200$ to 22.3% at $Re=2000$. The oscillation frequency $U^+$ for maximum DR slightly increases with $Re$, viz., from $U^+=0.06$ at $Re=200$ to 0.08 at $Re=2000$, with $DR_{max}=23.2%$. These results show that DR progressively decreases with increasing $Re$. Turbulent statistics and coherent structures are examined to explain the degradation of drag control effectiveness at high $Re$. FIK analysis in combination with the spanwise wavenumber spectrum of Reynolds stresses reveals that the decreased DR at higher $Re$ is due to the weakened effectiveness in suppressing the near wall large-scale turbulence, whose contribution continuously increases due to the enhanced modulation and penetration effect of the large-scale and very large-scale motions. Based on the power-law model and the log-law model, we predict more than 10% drag reduction at very high Reynolds numbers, say, $Re=10^5$.

9:29 AM Q26.00009 Constant power input simulations of drag reduced viscosity stratified turbulent channel flow, OSCAR NAZARENKO, TU Wien, ALESSIO ROCCON, TU Wien; University of Udine, FRANCESCO ZONTA, TU Wien; ALFREDO SOLDATI, TU Wien; University of Udine. In this work, we analyze the energy budgets of a turbulent channel flow in which a thin layer of fluid is used to lubricate the flow. In particular, we consider a setup in which a thin layer of fluid (viscosity $\eta_f$) is used to lubricate the flow of a thicker fluid layer (viscosity $\eta$). The system dynamics is investigated numerically using DNS of the Navier-Stokes equations coupled with a phase-field method. We consider a single-phase case and three multiphase cases. The multiphase cases differ by the value of the viscosity ratio employed: $\lambda=1.00$ (matched viscosity), $\lambda=0.50$ and $\lambda=0.25$ (less viscous lubricating layer). In order to obtain a more meaningful comparison among the different cases, simulations are performed using a Constant Power Input (CPI) framework and the power Reynolds number is kept fixed to $Re_{CPI}=12220$ (corresponding in the single-phase reference case to $Re_{ef}=300$). The results show that for all the cases considered, with respect to single-phase reference case, an increase of the thicker layer flow-rate is observed and Drag Reduction (DR) is obtained. The DR is linked to the interplay between inertial, viscous and surface tension forces which lead to an overall reduction of the turbulent dissipation.

Tuesday, November 26, 2019 7:45AM - 9:42AM — Session Q27 Flow Control: Model Reduction

7:45 AM Q27.00001 Energy-optimal phase control of unsteady fluid flows, ADITYA NAIR, University of Washington; KUNIHIKO Taira, University of California, Los Angeles; BINGNI Brunton, STEVEN BRUNTON, University of Washington. Controlling the phase of oscillation of unsteady fluid flows is crucial in terms of advancing and delaying flow field characteristics. Our goal is to design an optimal flow control strategy that alters the oscillation phase in fluid flows using minimum energy input. We perform a phase reduction analysis to construct a reduced-order model in terms of a phase-sensitivity function that tracks the phase of oscillation in response to impulse perturbations. Using the phase sensitivity function and its gradient, the optimal control law is obtained by solving the Euler-Lagrange equations as a two-point boundary value problem. We demonstrate the approach for incompressible flow over a cylinder at low Reynolds number and discuss extensions to complex flows. Multiple actuation strategies based on blowing and rotary control are also explored. Finally, we examine the synchronization and desynchronization properties of the optimally controlled flow and discuss broader implications for flow control.

7:58 AM Q27.00002 Spectral POD analysis of low Reynolds flow past finite cylinders, MATTEO CHIATTO, Department of Industrial Engineering, Università di Napoli Federico II, Italy; CAROLINE CARDINALE, JESSICA K. SHANG, Department of Mechanical Engineering, University of Rochester, NY, USA; LUIGI DE LUCA, Department of Industrial Engineering, Università di Napoli Federico II, Italy; FRANCESCO GRASSO, DynFluid Laboratory, CNAM- Arts et Metiers ParisTech, France. The need to understand the physics and to control a flow field has led scientists to develop techniques suitable for characterizing the dynamics and the topology of the flow based on simplified equations, and by exploiting experimental and/or numerical data. Major analysis techniques include POD, DMD, Spectral POD (SPOD, Towne et al. JFM 2018, 847). SPOD has the ability to represent structures evolving coherently in space and time, and it is here applied to study the flow past finite oblique cylinders. The study is aimed at investigating shedding regimes at various low Reynolds numbers. The wake behind the cylinder is experimentally investigated by means of a digital visualization technique described by Shang et al. (JFM 2018, 837). The major results consist in extracting the wake dynamics and identifying the eigen-directions associated with the most energetic structures. The wake field is reconstructed by considering just the few selected modes. Such a low dimensional decomposition is able to capture the dominant shedding modes, by comparing both image reconstruction and quantitative shedding Strouhal numbers to previous results.

8:11 AM Q27.00003 Model-Based Estimation of Vortex Sheding in Unsteady Cylinder Wakes, JIWEEN GONG, JASON MONTY, SIMON ILLINGWORTH, The University of Melbourne. This work considers single-sensor based estimation of vortex shedding in cylinder wakes at $Re=100$ in simulations and at $Re=1036$ in experiments. A model based on harmonic decomposition is developed to capture the periodic dynamics of vortex shedding. Two model-based methods are proposed to estimate time-resolved flow fields. First, Linear Estimation (LE) which implements a Kalman Filter to estimate the flow. Second, Linear-Trigonometric Estimation (LTE), which utilizes the same Kalman Filter together with a nonlinear relationship between harmonics of the vortex shedding frequency. LTE shows good performance and outperforms LE in reconstructing vortex shedding. Physically this suggests that, at the Reynolds numbers considered, the higher harmonic motions are slave to the fundamental frequency.

8:24 AM Q27.00004 The dynamical states of a circular cylinder wake influenced by a leeward control rod, MURILO CICOLIN, PETER BEARMAN, OLIVER BUXTON, Imperial College London; GUSTAVO ASSE, University of Sao Paulo. Experiments were carried out in a circulating water channel with a circular cylinder fitted with one control rod at different positions. The rod had a diameter 10 times smaller than the main cylinder. The distance between the centre of cylinders was kept constant (R/D=0.7) and the angle between them varied from 40 to 90 degrees, when at 0 degrees the control cylinder is placed in the wake on a line passing through the stagnation point of the main cylinder and its centre. Reynolds number was 20,000 for all cases, based on the main cylinders diameter. High-resolution PIV fields were acquired focusing on the two-dimensional wake of both cylinders. Results showed that for all cases the control rod induced a significant change in the wake when compared to the bare cylinder. Three main dynamical states were identified according to the position of control rod across the shear layer: immersed in the recirculation region, in the middle of shear layer or outside of it. Moreover, a bistable transitional state was identified between the first and second states. The second state shows a suppression of vortex shedding and a significant reduction of drag forces for the combined system.
8:37AM Q27.00005 Koopman mode representations of vortex dynamics in viscous flows\textsuperscript{1}. KE-CHU LEE, SAM KAUFMAN-MARTIN, SAMANEH SADRI, POORVA SHUKLA, IGOR MEZI, PAOLO LUZATTO-FEGIZ, University of California, Santa Barbara — Vortex dynamics plays an important role in transitional and turbulent flows, where instabilities introduce important criteria for safety and performance of systems like turbomachinery and aerospace vehicles. In order to control these vortical flows, accurate, general and efficient models of vortex dynamics are needed. Here we explore the ability of Koopman mode decomposition (KMD) to provide such models and uncover physical mechanisms. Using pseudo-spectral simulations, we consider vortex dynamics in viscous flow for different Reynolds numbers and different initial vortex geometries. We start with co-rotating vortex pairs, for which we find that KMD modes and eigenvalues are in close agreement with linear stability predictions. We find that the onset of symmetry-breaking is weakly sensitive to Reynolds number, but highly sensitive to small details in the initial conditions. We provide a detailed comparison of the relative abilities of KMD and proper orthogonal decomposition (POD) to capture this behavior. Finally, we consider implications for modeling and controlling more general vortex flows.

\textsuperscript{1}Funding: ARO MURI W911NF-17-1-0306

8:50AM Q27.00006 Dynamic mode analysis and control of vortical flows in pump sumps. BYUNGJIN AN, Ebara Corporation, QIONG LIU, University of California, Los Angeles, MOTOHIKO NOHMI, MASASHI OBUCHI, Ebara Corporation, KUNIHIKO TAIIRA, University of California, Los Angeles — Pump sumps are settling chambers for incoming flows prior to their removal by pumps. These pump sumps are widely used for drainage in pumping stations and power plants. During off-design operations, free surface and sub-surface vortices often appear in pump sumps, causing significant pump performance degradation and system vibrations. Modal analysis is performed to obtain insights to develop an effective and highly robust suppressing device of vortex for a scaled pump sump. Dynamic mode decomposition (DMD) is used to extract the dynamic features with respect to the vortical flow obtained from the large eddy simulation (LES). The dominant DMD modes are useful for understanding the complex vortex dynamics and developing a novel control device for the suppression of vortex formation. In addition to examining the complex flow in the pump sump, we also consider a wall-normal vortex model. Active flow control is considered for the attenuation of such vortex. The resulting increase in pressure at the vortex core suppresses the detrimental effects. A comparative analysis of steady and unsteady actuation methods is carried out through LES. The characteristics of the successful control mechanisms will be identified and discussed.

9:03AM Q27.00007 A Hybrid Low-order Model of Dynamic Lift Response to Time-varying Actuation\textsuperscript{1}. XUANHONG AN\textsuperscript{2}, Princeton University, DAVID WILLAMS\textsuperscript{3}, Illinois Institute of Technology — Time-varying actuation from a synthetic jet actuator has previously been shown to be an effective way of controlling unsteady flow separation. In order to integrate this type of actuation into a real-time control system, a low-order model of the lift response to the actuation is desired. The current work proposes a low-order approach to modeling the lift response to time-varying leading edge actuation on a stalled airfoil. Dynamic Mode Decomposition (DMD) is employed to extract from the flowfield the critical dynamical information, which is connected to the lift variation. It is shown that there are two sets of DMD modes associated with two different frequencies. These two frequencies are contained in both the actuation signal and the lift response. Based on the information provided by the critical DMD modes, we propose a hybrid low-order model which consists of a time delay decay model and a convolution integral model. This hybrid model is capable of accurately predicting the lift response of an airfoil to the time-varying actuation.

\textsuperscript{1}The support of AFSOR under Grant FA9550-18-1-0440 Numbers and ONR under Grant Number 00014-14-1-0533 is gratefully acknowledged.
\textsuperscript{2}Postdoctoral Research Associate
\textsuperscript{3}Professor

9:16AM Q27.00008 Modeling the transient response to momentum injection for flow over an airfoil in deep stall using data-driven and projection-based methods\textsuperscript{1}. KATHERINE ASZTALOS, SCOTT DAWSON, DAVID WILLIAMS, Illinois Institute of Technology — Direct numerical simulations are performed for leading-edge momentum injection control of flow over a NACA0009 airfoil at post-stall angles of attack. We consider cases where the wake is both stable (Re = 200) and unstable (Re = 500), and find that the flow response is comparable in both regimes, though with some sensitivity to the instantaneous flow state in the unstable case. We investigate the physical mechanisms governing the interactions between the disturbance and the natural flow state using both dynamic mode decomposition (which is an entirely data-driven approach), and Galerkin projection of the governing Navier-Stokes equations onto a subspace obtained from proper orthogonal decomposition. In particular, we show that projecting a linearized Galerkin projection model can accurately predict DMD modes, giving an understanding of the origin of these modes from the underlying governing equations. More generally, this also suggests a method for approximating DMD modes from non-time-resolved data. We assess the ability of the resulting models to predict the time-evolution of the flow state of the actuated system. We also study the ability of our models to capture the finite-time-horizon energy growth present in the full system.

\textsuperscript{1}The authors gratefully acknowledge the support for this work from the Achievement Rewards for College Scientists (ARCS) Foundation, Inc’s Scholar Illinois Chapter and the Illinois Space Grant Consortium (ISGC).

9:29AM Q27.00009 Effect of spanwise wall oscillations on dynamics and evolution of near-wall coherent structures in a turbulent pipe flow at low and moderate Reynolds numbers\textsuperscript{1}. DANIEL COXE, RONALD ADRIAN, YULIA PEET, Arizona State University — Presented is the temporal evolution of a single conditional hairpin vortex in a turbulent pipe flow with transversely oscillated walls at low and moderate Reynolds numbers, compared to a baseline non-oscillated turbulent pipe flow case. A conditional hairpin is generated by a Linear Stochastic Estimation of a fluctuating velocity field of a non-oscillated pipe flow, obtained via Direct Numerical Simulations with a spectral-element method. An extracted hairpin is placed as an initial condition into the flow of interest and is convected downstream by a turbulent mean velocity profile. A goal of the study is to investigate the effect of wall oscillations on the development and growth of a conditional hairpin and the mechanisms associated with a potential suppression of auto-generation in a drag reduced flow. Subsequent formation and evolution of secondary and tertiary hairpins and how these processes are modified by a spanwise wall oscillation are also documented. A spatially averaged Reynolds stress profile along with the wall shear stress are presented to quantify the effects of auto-generation.

\textsuperscript{1}This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number TG-ENG150019
8:24AM Q28.00004 Inertial particle distribution in high Reynolds number turbulence: wavelet-based scale-dependent statistics

KEIGO MATSUDA, Center for Earth Information Science and Technology, Japan Agency for Marine-Earth Science and Technology (JAMSTEC), Yokohama, Japan, KAI SCHNEIDER, Aix-Marseille University, CNRS and Centrale Marseille, Marseille, France, KATSUNORI YOSHIMATSU, Institute of Materials and Systems for Sustainability, Nagoya University, Nagoya, Japan — The nonlinear dynamics of inertial particles in high Reynolds number turbulence, and in particular particle clustering, are important fundamental processes in atmospheric science. Here we analyze particle data from three-dimensional direct numerical simulations of particle-laden homogeneous isotropic turbulence at high Reynolds number, using Voronoi tessellation of the particle positions, considering different Stokes numbers (St). The divergence of the particle velocity can be quantified by determining the volume change rate of the Voronoi cells. We show theoretically that for random particles in random flow the divergence satisfies a PDF of the ratio X/Y, where X and Y follow normal and 1’ distributions, respectively. For inertial particles we find that the PDF of the divergence deviates from the theoretical prediction. Joint PDFs of the divergence and the Voronoi cell volume illustrate that the divergence is most prominent in cluster regions and less pronounced in void regions. Moreover, the mean value of the divergence becomes negative inside the cluster regions for St ≥ 2, corresponding to convergence of inertial particles, while for large St the mean value turns to positive values as the Voronoi cell volume becomes smaller. Finally, we show that the divergence of the inertial particle velocity exhibits no correlation with the second invariant of the fluid velocity gradient tensor, which has some impact for modeling particle laden turbulence.

8:37AM Q28.00005 Life and death of inertial particle clusters in homogeneous turbulence

YUANQING LIU, LIAN SHEN, University of Minnesota, Twin Cities, REMI ZAMANSKY, University of Toulouse, FILIPPO COLETTI, University of Minnesota, Twin Cities — Although clustering is a widely observed phenomenon in particle-laden turbulence, our understanding of the formation, evolution, and destruction of particle clusters is still incomplete. Virtually all existing definitions of a cluster rely on the spatial coherence of the particle concentration field, neglecting its temporal persistence. The latter is in fact essential to the ability of the particles to interact with each other, and to modify the carrier fluid flow. Here we leverage simulations of homogeneous isotropic turbulence laden with small heavy particles, and develop a Lagrangian framework to follow them before, during, and after their time as part of a coherent cluster. We define a criterion to establish whether a cluster survives over successive time steps, and use it to characterize its lifetime. Moreover, we investigate the recurring features of the turbulence associated to the formation and destruction of a cluster. The impact of the lifetime definition on the results is also discussed.

1Visiting Scientist, Aix-Marseille University, CNRS, I2M, Marseille, France
8:50AM Q28.00006 Clustering of gas-solids flows in a vertical duct, AARON M. LATTANZI, SARAH BEETHAM, University of Michigan, KEE ÖNN FÖNG, FILIPPO COLLETTI, University of Minnesota, JESSE CAPECIELATRO, University of Michigan — In this work, numerical simulations of moderately dense gas-solids flows in the fully-developed region of a vertical duct are performed. The simulations are performed within a volume-filtered Eulerian-Lagrangian (EL) framework and compared to novel experimental measurements. The high mass loading considered here leads to significant two-way coupling and the spontaneous generation of densely-packed clusters that fall along the duct walls. Two-phase flow statistics are extracted from the simulations and compared against detailed experimental measurements obtained from high-speed imaging and particle-tracking velocimetry. Additionally, a model for the pseudo-turbulent Reynolds stress (PTRS) was implemented within the EL framework to account for sub-grid particle-induced velocity fluctuations. Simulations with the PTRS closure allow the effect of a pseudo-turbulence model on clustered flows to be rigorously assessed for the first time.

9:03AM Q28.00007 Highly concentrated falling inertial particles in a vertical duct/riser1, KEE ÖNN FÖNG, FILIPPO COLLETTI, Univ of Minnesota - Twin Cities — Highly concentrated particle-laden turbulent flows, such as flows found in fluidized beds and falling-particle receivers, form complex and poorly understood interactions owing to the strong feedback of the dispersed phase on the fluid and possible inter-particle collisions. We present experimental observations on the velocity response and topological distribution of highly concentrated, falling inertial particles in a vertical rectangular duct. The working fluid is air laden with size-selected glass particles. The experiment is conducted in two different configurations of free-falling particles, and particles suspended by flowing air, enabling particle volume fractions as high as 3E-2. Two different resolutions are employed - a full-scale view to capture large-scale motions of the particles and cluster formation using particle image velocimetry; and a zoomed-in view to resolve the individual motions of particles using particle tracking velocimetry. The findings are discussed in the context of collective effect of particles, the influence of clusters on the mean statistics, and the partitioning of particle velocities into spatially correlated and random uncorrelated motions.

1National Science Foundation, Army Research Office, MN Environmental and Natural Resources Trust Fund

9:16AM Q28.00008 Reynolds number dependence of heavy particle preferential concentration1, XIANGJUN WANG, MINPING WAN, Southern University of Science and Technology — The preferential concentration of particles is a classical problem in particle-laden turbulence. The dependence of particle clustering on Reynolds number is still an open question. Here, the Reynolds number dependence of heavy inertial particles in homogeneous isotropic turbulence has been investigated. According to the analysis of Voronoi tessellation, the preferential concentration of small heavy particles is studied with the increase of Taylor Reynolds number from 52 to 139, with the number density of particles fixed. There are two factors that determine the extent of particle clustering. The first one is the strength of vortices in turbulence; the other is the persistent time that characteristic vortices drive particles to preferentially accumulate during one turnover time. Obviously, a larger strength of vortices and a longer persistent time contribute to a more intensive preferential concentration of particles. It is uncovered that the strength of vortices increases as Reynolds number increases, whereas the persistent time decreases with Reynolds number. Consequently, the influence of the latter defeats that of the former in present investigation. Therefore, the degree of preferential concentration decreases with Reynolds number.

1NSFC Grant Nos. 91752201, 11672123, and 91752000; Grant No. JCYJ20170412151759222

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q29 Biological Fluids Dynamics: Large Vessels 611 - Fanette Chassagne, University of Washington

7:45AM Q29.00001 Hemodynamics Assessments of Ascending Thoracic Aortic Aneurysm – the Influence of Hematocrit with Fluid-Structure Interaction Analysis, HAN HUNG YEH, SIMON RABKIN, DANA GRECOV, The University of British Columbia — An aortic aneurysm is one of the cardiovascular diseases with localized abnormal growth of a blood vessel risking rupture or dissect. The precise pathological pathway for aneurysm progression and formation is not completely understood. In the current study, ascending thoracic aortic aneurysms (ATAA) are investigated using a fully coupled fluid-structure interaction method focusing on the changes in hematocrit under normotension and hypertension. Blood was modeled as a laminar incompressible flow using the Quemada model with varying hematocrits. The arterial wall anisotropy and hyperelasticity were considered. Given the influence of hematocrit on the degree of shear-thinning of blood, the current result could provide valuable information in clinical practice. Our results suggested that with the increase in hematocrit, the shear stress distribution and the maximum shear stress magnitude along the arterial wall would increase significantly. The wall stress distributions, however, remained unchanged with respect to the changes in hematocrit. In addition, the distribution of von Mises stress indicated that elevated stress under hypertension could depend on geometry. This geometrical influence in ATAA could be a risk factor predicting further aortic expansion.

7:58AM Q29.00002 The hidden role of wall shear stress in fluid mechanics of coronary artery atherosclerosis1, AMIRHOSSEIN ARZANI, MOSTAFA MAHMOUDI, Northern Arizona University — Wall shear stress (WSS) is arguably the most important parameter in the biomechanics of atherosclerosis. Traditionally, WSS is used to quantify the frictional force on the endothelial cells and study mechanotransduction. However, WSS also provides valuable information about near-wall transport. Biologically, near-wall transport of certain biochemicals and cells influence atherosclerosis progression. The recent concept of Langrangian WSS structures (WSS manifolds) provides a connection between WSS vectors and near-wall transport. In this talk, we demonstrate the connection between WSS and near-wall localization of several atherogenic and atheroprotective biochemicals and cells in coronary artery atherosclerosis. Finally, an automated computational framework developed in FEniCS is presented to study the two-way coupling between WSS and plaque growth in coronary arteries. A phenomenological WSS driven growth model is developed to simulate plaque growth, which in turn influences the blood flow and WSS patterns.

1Funding from the American Heart Association (19AIREA34430058) is acknowledged.
8:11AM Q29.00003 Optimization of saline flushing of the coronary artery via two-lumen catheter for visualization of chronic total occlusions. SYED FAISAL, ERIC SEIBEL, ALBERTO ALISEDA, University of Washington, Seattle, WA, USA — A Chronic Total Occlusion (CTO) is caused by atherosclerotic plaque that occludes blood flow in a coronary artery. To improve treatment outcomes, visualization of the occlusion and surrounding coronary artery wall supports diagnosis and guides treatment. We investigate fluid interaction between blood and saline from different catheter designs, to understand the dynamics of blood displacement and low-Reynolds number mixing, and to optimize the catheter operating parameters involved in flushing of blood from the arterial lumen between the catheter tip and the CTO. The pressures develop during the flushing process and the time to obtain optical transparency in the lumen are measured. Suction is introduced to influence the flushing process. The incorporated changes to the artery size, as well as the arterial pulsatile blood flow, along with changes in the relative position of the fluid injection and suction lumens. The plaque morphology is varied to mimic its build up in arteries, testing the viability of the catheter uses under a wide range of physiological and anatomical conditions. A novel method of saline injection, suction and control, which helps reduce risk of the catheter failure and avoids injury to the patient, is developed.

8:24AM Q29.00004 Quantifying pumping effort and peristaltic work using balloon dilation catheters. SHASHANK ACHARYA, Department of Mechanical Engineering, Northwestern University, WENJUN KOU, Gastroenterology Division, Feinberg School of Medicine, Northwestern University, SOURAV HALDER, Theoretical and Applied Mechanics, Northwestern University, JOHN E. PANDOLFINO, PETER J. KAHRILAS, Gastroenterology Division, Feinberg School of Medicine, Northwestern University, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — Balloon catheters are widely used to study the mechanical response of blood vessels as well as the active and passive response of tubular organs like the esophagus. In this work, we focus on the EndoFLIP, a device used to characterize the esophagus’ response to dilation. We present a simplified 1D model to predict the system’s response to peristalsis and then use it to generate work curves that show how the muscle energy is being spent. We also observe the development of different pumping regimes based on the operating parameters and present work curves for each of these patterns. An extension of this 1D model is used to enhance data obtained from the device and generate work curves for patients with abnormal peristalsis. The detailed characterization of this system presented here will aid in better interpreting the data obtained from balloon dilation catheters used in patients for any tubular organs or tissue vessels.

8:37AM Q29.00005 Spatiotemporal characterization of pulmonary hypertension under pulsatile flow conditions. NARASIMHA RAO PILLALAMARRI, SENOL PISKIN, ENDER FINOL. Department of Mechanical Engineering, University of Texas at San Antonio — Pulmonary hypertension (PH) is a progressive disease characterized by elevated pressure and vascular resistance in pulmonary arteries. Nearly one quarter of a million hospitalizations occur annually in the U.S. with PH as the primary or secondary condition. A definitive diagnosis of PH requires right heart catheterization (RHC) in addition to a chest computed tomography, a walking test, and others. RHC is invasive has inherent risks and contraindications. This study aims to investigate non-invasive surrogates for RHC measures. Pulsatile hemodynamic simulations were conducted in 28 PH patient-specific geometries with a flow solver developed by customizing OpenFOAM libraries (v5.0, The OpenFOAM Foundation). Quasi patient-specific boundary conditions were implemented using a Womersley inlet velocity profile and resistance outflow conditions. Hemodynamic indices such as wall shear stress (WSS), time-averaged WSS, oscillatory shear index, and blood damage index were evaluated along with clinical metrics such as pulmonary vascular resistance and compliance to assess possible spatiotemporal correlations. The results are promising in the context of a long-term goal of identifying computational biomarkers that can serve as surrogates for invasive diagnostic protocols of PH.

8:50AM Q29.00006 Acoustic trapping and collapsing of microbubbles in stents. FEIYAN CAI, FEI LI, Shenzhen Institutes of Advanced Technology, Chinese Academy of Sciences, PENGFEI ZHANG, Qiu Hospital of Shandong University, HAIRONG ZHENG, Shenzhen Institutes of Advanced Technology,Chinese Academy of Sciences — Stents are commonly used in a coronary artery or any artery after angioplasty, but has the drawback to cause arterial restenosis. Targeted drug delivery has proved their efficacy at preventing coronary restenosis near stents. Here we propose a microbubble-based approach to enhance drug delivery by ultrasonically exciting the elastic resonances of the stents to generate a sound field that can trap and collapse microbubbles carrying drugs, which are consequently released preventing coronary restenosis near stents. Here we propose a microbubble-based approach to enhance drug delivery by ultrasonically exciting the elastic resonances of the stents to generate a sound field that can trap and collapse microbubbles carrying drugs, which are consequently released.

9:03AM Q29.00007 Coronary Stenosis Diagnosing System Using 2-step Machine Learning Algorithm. YOUNG WOO KIM, SUSIE RYU, HEEMIN LEE, JOON SANG LEE, Yonsei University — In this study, a two-step machine learning (ML) algorithm is introduced to estimate fractional flow reserve (FFR) along with decision (DEC) for coronary artery. This paper suggests the possibility of ML-based FFR to overcome the computational fluid dynamics (CFD) based FFR estimation, which includes calculation time and accuracy. Both synthetic model and patient model are used for the training of 2-step algorithm. In the first step, synthetic models are analyzed by CFD method and are used to train Gaussian progress regression model in order to increase the quantity of data. In this process, not only FFR but also flow characteristics are considered. Patient models are used to train the second step of the algorithm, which is based on a support vector machine. This step provides more information related to biomechanical features from the patient. In result, both the calculation time and accuracy of the model were higher for ML-based FFR than that from CFD-based FFR. This study suggests that both flow characteristics and biomechanical features should be also included in the dataset of ML algorithm, by analyzing the weight factor of each features.

1NIH 5P01DK117824-02 (Role of altered response to Volumetric Distension in Esophageal disease)

3This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (NRF-2017M3A9E9073371).

2Membership pending

4Corresponding author
reveals that there exists a constant force plateau for a narrowing of the ascending aorta and the stenosed area is subjected to abnormal hemodynamics and pressure drop. The pressure gradient across the stenosed region indicates the energy loss of the blood flow and is indicative of the extra work for the muscle of the left ventricle. The goal of this work is to develop a novel modeling approach to determine the accurate prediction of the pressure drop across the stenotic. We develop a computational method to simulate the propagation of pressure waves and the related arterial wall deformation. Blood is modeled using Navier-Stokes equations and the arterial wall consists of a thick material which accounts for the media and adventitia. Model simulations of the aortic blood flow under physiological conditions were performed using finite element method. The energy estimation is derived from weak formulation and applied to the stenosed artery to assess energy loss. The numerical simulations investigate the relation between the flow condition, pressure gradient and wall deformation for different levels of stenosis to cover the broad range of the degree of narrowing, ranging from trivial to severe.

Comparisons of the predictions coming from the second order theory are provided in order to critically assess its validity and usefulness. We find that the second order equations recover many of the anomalous features (e.g., non-constant pressure and non-zero parallel heat flux). Monte Carlo (DSMC) method have shown that at small Knudsen number the pressure and temperature profiles in the thermal stress problem do not provide accurate results. Second order equations were derived and shown to reproduce experimental results of the shock structure of gases over a large range of Mach numbers (Paolucci & Paolucci JFM, 486, 686-710 (2018)).

We apply mDMDc to patient-specific computational fluid dynamics data (velocity and wall shear stress vectors) in cerebral aneurysm and coronary artery stenosis models. Reconstruction errors are presented and different modes are compared between velocity and wall shear stress.

Tuesday, November 26, 2019 7:45AM - 9:42AM
Session Q30 General Fluid Dynamics 612 - Samuel Paolucci, Notre Dame

7:45AM Q30.00001 Surface morphology and flow dynamics for fog harvesting , FAN KIAT CHAN, University of Illinois, Urbana-Champaign, AIDA SHAHROKHIAN, HUNTER KING, University of Akron, MATTIA GAZZOLA, University of Illinois, Urbana-Champaign — Harvesting fog as a source of fresh water is a practical solution for inhabitants of foggy coastal deserts, whether human, animal or plant. Namib desert beetles famously lean their textured bodies into the fog-laden wind: their evolved strategy has been interpreted mostly in the context of surface wetting and its role in transporting of water as it is accumulated. However, little is known about how the interplay between flow dynamics and surface morphology affects droplet interception. From experiments with controlled flow and fog delivery and sensitive accumulation measurements as well as complementary numerical flow simulations, we find mechanisms for significant enhancement of collection efficiency on two scales: millimetric surface features which affect droplet trajectories; and microscopic features which aid evaporation of lubricating film before contact.

7:58AM Q30.00002 Simple Flows Using a Second Order Theory of Fluids , SAMUEL PAOLUCCI, University of Notre Dame — The Navier-Stokes-Fourier (NSF) equations have proved very valuable in modeling fluid flows over the last two centuries. However, there are cases in which large gradients in velocity and/or thermal fields occur where it has shown that they do not provide accurate results are divided into different phases of the cardiac cycle and DMD with control is applied to each phase. We apply mDMDc: to patient-specific computational fluid dynamics data (velocity and wall shear stress vectors) in cerebral aneurysm and coronary artery stenosis models. Reconstruction errors are presented and different modes are compared between velocity and wall shear stress.

8:11AM Q30.00003 Can we always neglect the bulk viscous pressure in variable density flows? , MILTIADIS PAPALEXANDRIS, Universite catholique de Louvain — In linearly isotropic (Newtonian) fluids, the bulk viscous pressure is proportional to the velocity divergence, with the bulk viscosity of the fluid being the proportionality coefficient. Stokes’ hypothesis states that the bulk viscosity of a Newtonian fluid can be set equal to zero. Although not valid for many fluids, it is common practice to invoke this hypothesis in the study of low-Mach-number, variable-density flows. In this talk, based on simple scaling arguments we provide a necessary condition for neglecting the bulk viscous pressure from the governing equations of low-Mach number flows. More specifically, we show that the Reynolds number defined with respect to the bulk viscosity must be very large. We further show that even when this condition is satisfied, the bulk viscous pressure does not need to be taken explicitly into account because it can be combined with the dynamic pressure.

8:24AM Q30.00004 New Memory Effects in Unsteady Forces on Slip Particles , HSIEH-HUNG WEL, A.R. PREMLATA, National Cheng Kung University — For a slip particle moving unsteadily in a viscous fluid, the extent of slip is not fixed but constantly varies with time, depending on the slip length λ relative to the boundary layer thickness δ. For this dynamic slip situation, we show that even the amount of slip is minuscule, it can significantly change the characteristics of the history force. Our analysis reveals that there exists a constant force plateau for δ<δ, persisting until the slip-stick transition point δ ~ Δ after which the usual no-slip Basset 1/δ decay dominates the force response. We also analyze the unsteady force response for a stick-slip Janus particle, showing that a slightly smaller Basset force can re-emerge in place of the slip force plateau in the small δ regime. This re-entrant history force transition becomes even more evident for a slip particle having a small no-slip patch. These unusual force responses may not only provide unique hydrodynamic fingerprints for characterizing heterogeneous particles, but also have potential uses in active manipulation and sorting of these particles.
8:37AM Q30.00005 Exact Navier-Stokes solutions linear in one coordinate. JONATHAN MESTEL, Mathematics Dept, Imperial College London — If a 3D flow is independent of one coordinate it naturally reduces to a 2D flow. Similar simplification can occur if a flow varies linearly with a coordinate. For example, the advection-diffusion equation \( \nabla \cdot u = \kappa \nabla^2 c \) has solutions of the form \( c = xf(y, z) \) when the velocity has the form \( u = (xu(y, z), v(y, z), w(y, z)) \) with \( \nabla \cdot u = 0 \). The resulting system is essentially two-dimensional, but retains some 3D aspects. This talk employs similar reductions in axisymmetry to derive several previously unknown solutions to the full Navier-Stokes equations. As they extend to infinity, in some cases these similarity solutions exist without additional forcing. A family of 3D boundary layer flows is also derived, demonstrating for example that the Falkner-Skan solutions are nonunique in 3D. Finally, it is shown that these flows can coexist with other fields of advection-diffusion type. In particular, it is shown that these flows can act as dynamos, spontaneously generating magnetic fields with a related spatial structure.

8:50AM Q30.00006 Sedimentation of Triple Twisted Möbius Objects. NICOLAS MORENO, DAVID VASQUES-CORTES, MICHAEL GRUNWALD, JOHANNES SCHOEENKE, ELIOT FRIED, Okinawa Institute of Science and Technology — Screw-like objects like helic and Möbius strips exhibit both rotation and translation as they move in a fluid due to external forces such as gravity. It is expected that the orientation of the twist in the structure determines the direction of the rotational motion. Here, we conduct both computational and experimental studies on the sedimentation of rigid Möbius bands with a three-half twist. We vary the aspect ratio of the bands and analyze their trajectories. We use two different schemes to construct the bands corresponding ruled developable surfaces and ruled binormal-scroll surfaces. For developable bands, we observe that the spinning direction is consistently determined by the orientation of the band and independent of its aspect ratio. Remarkably, for binormal-scroll bands, the direction of spin depends not only on the twist orientation but also on aspect ratio. Experimentally, we track the sedimentation of 3D-printed polystyrene bands in water. Computationally, we use the particle-base method dissipative particle dynamics. The hydrodynamic properties of these objects offer exciting applications for mixing and separation in microfluidics and may also serve as prototypes for passive swimmers.

9:03AM Q30.00007 ABSTRACT WITHDRAWN —

9:16AM Q30.00008 Lubricated Rolling Over A Pool. HATEF RAHMANI, SHELDON GREEN, BORIS STOEBER, Department of Mechanical Engineering, University of British Columbia, NEIL BALMFORTH, Department of Mathematics, University of British Columbia — Film splitting flows are important in many industrial processes, such as the coating of railroad tracks with liquid friction modifiers (LFM). Here, an LFM pool deposited on the track is overridden by an advancing wheel; a liquid film coats both the wheel and rail surfaces, splitting at a meniscus behind the wheel. Lubrication theory is used to predict the meniscus position, wetted length and pressure on the wheel as a function of the peak-to-valley height of the rail, two solutions exist: below a critical load, steady planing arises in which the minimum gap and wetted length are adjusted to match the incoming and outgoing fluid fluxes. Above that load, flooding occurs with a steady minimum gap, but the incoming flux exceeds that underneath, and fluid is ploughed before the wheel. In 3D, only planing is possible, with leakage to the sides eliminating flooding states. Laser-induced fluorescence and high-speed imaging measure coated film thicknesses in experiments that are within 19% of model predictions.

9:29AM Q30.00009 Understanding the behavior of large scale structures in highly scalable Spectral element simulations of Internal Combustion Engines in the framework of large eddy simulation. TANMOY CHATTERJEE, SAUMIL PATEL, MUHSIN AMEEN, Argonne National Laboratory — In this talk, we investigate the large scale structures generated in various flow regimes (intake, compression, expansion, and exhaust strokes) of a motored, TCC-III internal combustion (IC) engine. The study is performed with a highly scalable spectral element code Nek5000. The moving boundary problem corresponding to the valves and piston motion are incorporated through a spectrally accurate arbitrary Lagrangian-Eulerian methodology. To avoid severe mesh distortions, we use a novel methodology of interpolating simulation data to a redesigned mesh in situ, thereby keeping the shape of the meshes in control during the simulations. We also use a novel overset-grid methodology to mesh the spark plug region separately in order to reduce re-meshing of the spark region and maintain stability in our simulations. Such high fidelity studies are not only crucial for understanding the multiscale engine-flow dynamics and cyclic variability of the flow features, but also serves as an efficient guideline for developing more advanced models, like species reaction, and spray injection corresponding to realistic IC engines.

Tuesday, November 26, 2019 7:45AM - 9:42AM —

Session Q31 Biological Fluid Dynamics: Locomotion Cilia 613 - David Saintillan, University of California San Diego

7:45AM Q31.00001 Hydrodynamic synchronization of spontaneously beating filaments. DAVID SAINTILLAN, BRATO CHAKRABARTI, University of California San Diego — Cilia and flagella are thin hair-like cellular projections that play a range of crucial roles from propulsion at low Reynolds number to long-range hydrodynamic transport. The movement of the cilia is produced by the bending of its core, known as the axoneme, consisting of 9+2 pairs of microtubules. In presence of ATP, molecular motors connecting microtubules undergo cycles of attachment and detachment and generate sliding forces that are converted to wavelike motion of the filaments. We present a microscopic model that accounts for the stochastic kinetics of molecular motors and mechanical feedback from the axoneme, and produces spontaneous oscillations consistent with those of sperm, cilia and Chlamydomonas. Using this model for the axoneme, we study elastohydrodynamic phase synchronization in a pair of beating filaments. Our simulations reveal that symmetric sperm-like beats lead to in-phase synchrony while both in-phase or anti-phase synchrony can emerge for asymmetric ciliary waveforms. We find that phase-synchronization is well captured by a low-dimensional Adler equation and also elucidate the role of biochemical noise in driving phase slips.

7:58AM Q31.00002 Dynamics of a cilia/cilium beating in 3D non-Newtonian flow. CHENGLEI WANG, SIMON GSELL, UMBERTO D’ORTONA, JULIEN FAVIER, Aix-Marseille Univ, CNRS, Centrale Marseille, M2P2, Marseille, France — Cilia are micro-scale hair-like organelles protruding from the surfaces of eukaryotic cells. Through fluid-structure interaction (FSI), they usually serve for fluid transport and locomotion. Such a FSI problem has been widely explored recently. In most existing works, the fluid is modeled as Newtonian. However, this is not always the case in nature, such as for the airway surface liquid (ASI) covering the epithelial surface of the respiratory system of the human body. In other words, the non-Newtonian flow could play a significant role on the cilia dynamics, which yet has been rarely studied. Therefore, this study aims to bridge this gap. Specifically, the non-Newtonian fluid is described using the power-law model, and each cilium is represented by a flexible filament. A single cilium or an array of cilia are placed in the fluid and driven at their base by a configuration-dependent torque. With a well-established numerical solver based on the immersed boundary lattice Boltzmann method (IBLBM) and the nonlinear finite element method (FEM), the cilia dynamics and their hydrodynamic interactions in the 3D non-Newtonian flow are systematically investigated, and the effects of several key parameters including the power-law index and the cilia spacing are also revealed.
8:11AM Q31.00003 The role of flexibility in sub-inertial swimming: An analysis of millimeter-scale ciliated structures1, ADRIAN HERRERA-AMAYA, Department of Mechanical Engineering, The Pennsylvania State University, FERHAT KARAKAS, DAVID W. MURPHY, Department of Mechanical Engineering, University of South Florida, MARGARET L. BYRON, Department of Mechanical Engineering, The Pennsylvania State University — Ctenes are rectangular paddle-like structures approximately one millimeter in length, composed of packed bundles of very long cilia. They are used by ctenophores (gelatinous marine zooplankton) to swim at intermediate Reynolds numbers (Re). Using Particle Shadow Velocimetry (PSV), we experimentally examine variations in the beat kinematics and fluid dynamics across ontogeny of the ctenophore Bolinopsis vitrea. In smaller animals, the cten kinematics resemble the physics of micro-cilia, using spatial asymmetry to produce net thrust. By contrast, the ctenes of larger animals show a decreased spatial asymmetry but maintain the overall thrust by increasing the velocity difference between power and recovery strokes (a strategy which would be ineffective in low Re, time-reversible flows). We also observe wall-normal displacement of the mesoglia which is detectable, nontrivial, and with a frequency approximately equal to that of the cten beating. Overall, we show that flexibility is a major parameter in ctenophore swimming, and that further study of this system could help elucidate the physics of flexible propulsors at the boundary of the viscous and inertial regimes.

1The National Academies Keck Futures Initiative, and CONACYT Graduate Fellowship

8:24AM Q31.00004 Long-range self-organization of ciliary activity and flow patterns in reconstituted bronchial epithelium1, ANNIE VIALLAT, Aix Marseille University, CNRS, ETIENNE LOISEAU, SIMON GSELL, UMBERTO D’ORTONA, JULIEN FAVIER, Aix marseille university, CINAM TEAM, M2P2 TEAM — Micociliary clearance is the active transport of a complex fluid, mucus, along the airway epithelial surface. Mucus is propelled over tens of centimeters by the beating of billions of active cilia carried by the epithelial ciliated cells. How the necessary coordination of beat directions emerges during ciliogenesis and is maintained is still an open debate. Would the collective motions of ciliary beats involve the dynamics interaction between cilia as observed in long range interaction in active matter systems? The direction of ciliary beats is constrained by the long-range hydrodynamic forces created by distant cilia and mediated by mucus, and by the planar polarity of the tissue. Here, after highlighting the spontaneous emergence and growth of mucus swirls during ciliogenesis, we show that mucus is necessary to generate and maintain a global swirl, associated with a strong circular directional order of ciliary beats, spanning the whole culture. By showing that large-scale swirl and ciliary order are lost and then recovered by successively removing and adding mucus to the epithelial surface, we demonstrate that the hydrodynamic force exerted locally on each cilium by the mucus flow, itself resulting from the beats of all the cilia of the epithelium, induces its active reorientation. These results are discussed in light of a hydrodynamic model which captures the observed mucus flow patterns.

1support:A*MIDEX funded by French Government

8:37AM Q31.00005 FAST, NEAREST and Flagellar Regulation1, MEURIG GALLAGHER, GEMMA CUPPLES, JACKSON KIRKMAN-BROWN, DAVID SMITH, University of Birmingham — In an age where huge amounts of data can be readily produced it is increasingly important to be able to accurately and efficiently analyse large amounts of data, and to be able to use these analyses as a marker for clinical outcome. However, semen analysis in the human is currently limited to methods such as sperm counting and analysis of fixed cells. To address this, we have developed and released FAST, a free-to-use package for the high-throughput detection and tracking of large numbers of beating flagella in experimental microscopy videos. This new ability to track the detailed flagellar waveform allows for more than just measurements of motility. Alongside FAST we have been developing NEAREST, an open source software enabling the rapid application of a meshless regularised Stokeslet method to solve mobility and resistance problems in Stokes flow. Combining FAST and NEAREST allows for detailed investigation into experimentally intractable quantities such as energy dissipation, disturbance of the surrounding medium and viscous stresses. Finally, we will discuss how the analysis capabilities provided by these tools can be combined, together with models of the elastic behaviour of flagella and learning algorithms to probe the secrets of flagellar regulation in swimming cells.


8:50AM Q31.00006 Modeling the Synchronization of Flagella on the Exterior of a Sphere, KARIN LEIDERMAN, Colorado School of Mines, FOREST MANNAN, Western Colorado University, MIKA JARVELA, Colorado School of Mines — Flagella are hair-like appendages attached to microorganisms that allow the organisms to traverse their fluid environment. The algae Volvox are phototactic, spherical swimmers with thousands of biflagellate somatic cells embedded in their surface. Their flagella coordinate their beating, which leads to forward swimming with slight rotations; the coordination of their flagella is not fully understood. In this work, we extend a previously published mathematical model of coordinated flagella on a flat surface to study coordination on the exterior surface of a sphere. The goal was to determine if factors related to the spherical shape affected flagellar synchronisation. Each beating flagella itself is modelled as a small rotating sphere, attached to a point just above the spherical surface by a spring, and the effects of all other flagella are accounted for with a regularized image system for Stokes flow outside of a sphere. We consider flagellar beating in meridional planes with slight offsets and from the anterior to posterior pole. We found that this minimal model can achieve large-scale coordination of flagella that leads to metachronal waves and also results in velocity fields that represent forward swimming with rotation. We varied parameters for meridional offsets, spring stiffness and number of flagella to study how they each affected the coordination. This study lays the groundwork for future studies to better understand their more complex coordination needed to drive their phototactic behavior.

9:03AM Q31.00007 A computational study of amoeboid motility in 3D: Role of extracellular matrix geometry, cell deformability and cell-matrix adhesion1, PROSENJIT BAGCHI, ERIC CAMPBELL, Rutgers University — Cells exhibiting amoeboid locomotion are abundant within the human body, as immune cells, epithelial cells, neuronal cells, embryonic cells, and even metastatic cancer cells migrate using the amoeboid phenotype. Amoeboid locomotion is accomplished through the use of pseudopods, or cylindrical membrane extensions which protrude, bifurcate, and retract dynamically, resulting in a net cell displacement. The modeling of amoeboid locomotion is a complex and multiscale process, where large cell deformation, protein biochemistry, and both cytosolic and extracellular fluid interactions must be considered. Furthermore, cells are often confined inside the extracellular matrix (ECM), a porous, fluid-filled medium. Adhesive interactions between the cell and underlying substrate add further layers of complexity. In this work, we present a 3D computational model of amoeboid migration in various ECM geometries. Our model couples a fluid/structure interaction for extreme cell deformation, a pseudopod generating activator-inhibitor system, cytoplasmic and extracellular fluid motion, and a fully resolved extracellular matrix. Simulation results show a strong coupling between cell deformability, matrix geometry and cell-ECM adhesion providing valuable on the mechanics of amoeboid migration.

1Funded by National Science Foundation
9:16AM Q31.00008 Efficient Implementation of Elastohydrodynamic Integral Operators for Stokesian Filaments. ATTICUS HALL-MCNAIR, THOMAS MONTÉNEGRO-JOHNSON, University of Birmingham, HERMES GADLHA, University of Bristol, DAVID SMITH, MEURIG GALLAGHER, University of Birmingham — Models for simulating the dynamics of flexible biological filaments have historically been mathematically complex and numerically expensive, in part due to numerical stiffness associated with satisfying fiber inextensibility. Moreau et al. (2018) recently demonstrated that a filament could be modeled efficiently by expressing the governing elastohydrodynamic problem via integral equations written in terms of the evolving tangent angle. Combined with a method of lines discretization, the required degrees of freedom for accurate simulation are reduced, alleviating numerical stiffness and enabling efficient computational simulation. In this presentation, I outline recent work which builds upon the Moreau et al. framework, augmenting their formulation with the method of regularized stokeslets of Cortez et al. (2001, 2005) to enable the modeling of non-local hydrodynamic interactions within and between filaments. From this, the non-linear filament deformations caused by shear flows and sedimentation are modeled efficiently, revealing how fiber interactions can lead to geometric buckling instabilities. We also consider the dynamics of active moment driven swimmers for different moment types, and investigate optimal parameter pairings to produce fast swimming in active filaments.

9:29AM Q31.00009 Magnetically powered metachronal waves induce locomotion in capillary self-assemblies. YLONA COLLARD, GALIEN GROSJEAN, NICOLAS VANDEWAELLE, University of Liege — When tiny soft-ferromagnetic particles are placed along a liquid interface and exposed to a vertical magnetic field, the balance between capillary attraction and magnetic repulsion leads to self-organization into well-defined patterns. We demonstrate that precessing magnetic fields induce metachronal waves on the periphery of these assemblies, similar to the ones observed in ciliates and some arthropods. The outermost layer of particles behaves like an array of cilia or legs whose sequential movement causes a net and controllable locomotion. This bioinspired many-particle swimming strategy is effective even at low Reynolds number, using only spatially uniform fields to generate the waves. The motivations for studying these systems range from the fundamental understanding of biological processes to the development of medical and technological applications.

Tuesday, November 26, 2019 7:45AM - 9:29AM – Session Q32 Biological Fluid Dynamics: Vesicle and Micelles II

7:45AM Q32.00001 Transport phenomena in a fluid film with curvature elasticity. ARJIT MAHAPATRA, DAVID SAINTILLAN, PADMINI RANGAMANI, Department of Mechanical and Aerospace Engineering, University of California San Diego — Lipid bilayers are fluid films that are elastic in bending. Cellular membranes are lipid bilayers that contain different proteins, including ion channels, receptors, and scaffolding proteins. These proteins are known to diffuse in the plane of the membrane and to influence bending of the membrane. Experiments have shown that lipid flow in the plane of the membrane is closely coupled with the diffusion of membrane proteins. Thus there is a need for a comprehensive framework that accounts for the coupling between these processes. Here, we present a theory for the coupled in-plane viscous flow of lipids, diffusion of membrane proteins, and curvature elastic deformation of lipid bilayers. The proteins in the membrane are modeled such that they influence membrane bending by inducing a spontaneous curvature. We formulate the free energy for the system as a Helfrich-like curvature elastic energy density function modified to account for the chemical potential energy of proteins. Then, we apply the principle of virtual work to minimize the free energy and derive conservation laws and equation of motions. Finally, we present results from dimensional analysis and numerical simulations and demonstrate that asymmetry in protein distribution plays an important role in driving lipid flows.

7:58AM Q32.00002 Formation of vesicles in a viscous solvent: A hybrid coarse-grain/continuum approach. SZU-PEI FU, ROLF RYHAM, Fordham University, YUAN-NAN YOUNG, New Jersey Institute of Technology — In this talk a theoretical model for long-range, hydrophobic attraction between amphiphilic particles (such as lipid molecules) is developed to quantify the macroscopic assembly and mechanics of a lipid bilayer membrane in solvents. The non-local interactions between amphiphilic particles are obtained from the first domain variation of a hydrophobicity functional, giving rise to forces and torques (between particles) that dictate the motion of both particles and the fluid flow in the viscous solvent. Both the hydrophobic and hydrodynamic interactions between the coarse-grained amphiphilic particles are formulated into integral equations, which allow for accurate and efficient numerical simulations in two- and three-dimensions. Such hybrid coarse-grained model is validated by its capability to reproduce various physical properties of a lipid bilayer membrane. Furthermore we present simulation results of vesicle formation in both a quiescent flow and a pressure driven flow (as in the microfluidic jetting experiments). Finally we also illustrate how our hybrid model can be generalized to investigate the effects of local charges on the bending rigidity of a lipid bilayer membrane.

8:11AM Q32.00003 Hydrodynamic shear transmission across cellular membranes. DANIEL TAM, GUILLERMO AMADOR, MARIE-EVE AUBIN-TAM, TU Delft — A cell’s interactions with the environment are mediated by its cellular membrane. This nanometer-thick, liquid crystalline structure is mostly composed of a lipid bilayer, which serves as a scaffold for embedded proteins and other macromolecules. Many crucial cellular processes, such as growth, proliferation, and protein complex formation, are dependent on external mechanical stresses; therefore, understanding how cellular membranes generate and transmit forces may shed light on cell behavior and homeostasis. In this study, we use optical tweezers to both apply and measure local forces on free-standing lipid bilayers within microfluidic channels. The planar geometry of the lipid bilayer facilitates interpretation of measurements using hydrodynamic models. This technique is the first to combine multiple optical tweezers probes with planar free-standing lipid bilayers accessible on both sides of the bilayer. The aims of these measurements are to quantify fluid slip close to and transmission of shear forces across the bilayer surface, building towards a fundamental understanding of the physical principles governing the transfer of forces by and through the membrane.

8:24AM Q32.00004 Electric Field-driven Deformation and Translation of Vesicles in Microchannels. ADNAN MORSHEDE, PRASHANTA DUTT, Washington State University — Bio-mimetic vesicles like liposomes exhibit complex responses when exposed to electric pulses. Their lipid membranes bear close resemblance to biological cells. Yet the lack of internal cytoskeletons and transmembrane organelles lead to different membrane deformation characteristics. The response of these vesicles is also affected by the electrical properties of the media and vesicle size and shape. We investigated the electrodeformation and transport of bio-mimetic vesicles immersed in a fluid media under a DC electric field. The deformation characteristics of vesicle membrane was represented with a Mooney-Rivlin constitutive law. The electric field, flow field, and vesicle deformation are resolved with a hybrid immersed resolved technique. Additionally, electroosmotic flows are considered for a range of surface charge conditions. Depending on the direction and magnitude of the electroosmotic flow, tumbling, tank treading, and pure translational motions were observed. Furthermore, a logarithmic scaling is realized between the stretch ratio of the vesicle and shear rate under electroosmotic flows. Conductivity ratio of vesicle and the surrounding media is found as a key parameter in the translational motion as well.

ONR N00014-17-1-2628
8:37AM Q32.00005 Active particle penetration through a planar elastic membrane¹.

ABDALLAH DADDI-MOUSSA-IDER, BENNO LIEBCHEN, ANDREAS M. MENZEL, HARTMUT LOEWEN, Institut fuer Theoretische Physik II: Weiche Materie, Heinrich-Heine-Universitaet Duesseldorf, Germany — Active penetration of nanoparticles through cell membranes is a fascinating phenomenon that may have important implications in various biomedical and clinical applications. Using particle-based computer simulations and theory, the penetration mechanism of an active particle through a planar elastic membrane is studied. The membrane is modeled as a self-assembled sheet of particles embedded in a Newtonian viscous fluid. A coarse-grained model is introduced to describe the mutual interactions between the membrane particles. Three distinct scenarios are identified, including trapping of the active particle, penetration through the membrane with subsequent self-healing, in addition to penetration with permanent disruption of the membrane. The latter scenario may be accompanied by a partial fragmentation of the membrane into bunches of isolated or clustered particles. Our approach might be helpful for the prediction of the transition threshold between the trapped and penetrated states in real-space experiments involving motile swimming bacteria or artificial active particles. Reference: A. Daddi-Moussa-Ider, B. Liebchen, A. M. Menzel, and H. Lwen, Theory of active particle penetration through a planar elastic membrane, New J. Phys. in press (2019).

¹The authors gratefully acknowledge support from the DFG.

8:50AM Q32.00006 Viscoelasticity of a bacterial extracellular polymeric substance (EPS) streamer filament using micro-rheology and microfluidics, ANDREW WHITE, MARYAM JALALI, JIAN SHENG, Texas A&M — Using Ecology-on-a-chip (eChip), we have demonstrated that polymeric aggregates can be formed around a rising oil micro-droplet by Pseudomonas. The EPS aggregate is initiated by forming trailing streamers with one end anchoring at the droplet surface and the other floating in the flow, which alters “wake” pressure field and consequently causes substantial drag on the streamers and their surrounding shear flows, viscoelastic behavior of streamers must be understood. Here, we apply micro-rheology technique to quantify real-time viscoelasticity of streamers developed in a pinned oil droplet in eChip. Using high speed microscopy, filament strain is determined by tracking trapped bacteria in real-time and concurrently viscous stresses are measured using PIV-assisted PTV of freely suspended bacteria. Stress-strain shows hysteresis of viscoelastic materials. Funded by GoMRI, ARO

9:03AM Q32.00007 Species and orientation differences in copepod behavior in a Burgers vortex, D. ELMI, S. SOUMYA, D.R. WEBSTER, Georgia Tech, D.M. FIELDS, Bigelow Laboratory for Ocean Sciences — We use a Burgers vortex experimental model to study the interaction of two marine copepod species (Temora longicornis and Acartia tonsa) with a single turbulent-like eddy structure. Tomographic-PIV experiments were performed at small-scale to quantify the velocity field of turbulent vortices modeling those that copepods encounter in their oceanic habitat. Turbulence intensities are discretized into four levels corresponding to dissipation rates of 0.002 to 0.25 cm²/s³. Three-dimensional swimming trajectories are retrieved from two orthogonal camera perspectives and overlaid on the Burgers vortex velocity structure to identify individual swimming kinematics and behavioral differences. We use apparatuses with the vortex axis aligned in horizontal and vertical directions to argue that the copepods are sensitive to the directionality of the hydrodynamic signal. By comparing the behavioral change in an observed flow field, the Pirouette-Driver or control treatment, we provide a framework to identify trends in behavioral responses. Both species of copepods increase their relative velocity as the strength of the Burgers vortex increases and the trend appears stronger in the vertically-aligned vortex. Further, the number of circular-shaped trajectories around the vortex core increases with increasing strength of the Burgers vortex for both species, and the trend is stronger for A. tonsa. These trends, together with the statistical analysis, suggest that changes in oceanic turbulence intensity as a result of climate change would affect the distribution of copepods.

9:16AM Q32.00008 An experimental implementation of a two-sphere swimmer at low Reynolds numbers, OLIVER SILVERBERG, BRENT HOSOUME, NIKHIL TRIVEDI, CONNOR TISCH, Mechanical Engineering, Santa Clara University, DANIEL PLASCENCIA, Bioengineering, Santa Clara University, MATTHEW HOLMES, ON SHUN PAK, Mechanical Engineering, Santa Clara University, LORICATION AT LOW REYNOLDS NUMBERS encounters stringent constraints due to the dominance of viscous over inertial forces. Various elegant designs have been proposed to escape from the constraints of the scalar theorem and generate self-propulsion. In this talk, we present a macroscopic experimental implementation of the PushmePullyou swimmer (J. E. Avron, O. Kenneth, D. H. Oaknin, New J. Phys., 7, 234, 2005), which consists of a pair of expandable spheres connected by an extensible link. We characterized the propulsion performance of the swimmer in the low Reynolds number regime with the use of highly viscous silicone oil and compared the results with theoretical predictions.

Tuesday, November 26, 2019 7:45AM - 9:29AM

Session Q33 Flow Instability: Control 615 - Mathew Juniper, University of Cambridge

7:45AM Q33.00001 Shape Optimization for Stability of a Cyclone Separator, MATTHEW JUNIPER, JACK BREWSTER, University of Cambridge — A cyclone separator uses a swirling flow to remove particles from a particle-laden fluid. Cyclone separators are frequently used in domestic applications and in industries for clean contamination and in industries for gases. At practical Reynolds and swirl numbers, the steady axisymmetric flow through a cyclone becomes linearly unstable. It develops a precessing vortex core (PVC) which is responsible for increased pressure loss and unwanted acoustic noise. Unsteady mixing caused by the precession also leads to the re-entrainment of separated particles. We perform shape optimization of a cyclone separator in order to weaken this instability. The onset of the PVC appears as an unstable global mode with azimuthal wavenumber m = 1. We calculate the shape gradient of the growth rate of this global mode. We identify the boundary regions that most influence the growth-rate and then use a gradient-based method to update and optimize the geometry. We interpret this physically and also present a family of orthogonal geometry changes that cause the greatest changes in the cyclones base-flow. Theis process highlights the geometry changes that a parametrisation must be able to reproduce in order to effectively optimize a cyclone.

7:58AM Q33.00002 Control of Laminar-turbulent transition on a natural laminar flow airfoil, JOHN WYLIE, KEITH TAYLOR, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Tollmen-Schlichting (T-S) waves are the principal mechanism for laminar- turbulent transition over external surfaces. The present work aims to identify T-S waves on a natural laminar flow airfoil in the presence of an adverse pressure gradient, and subsequently reduce or eliminate the amplitude of identified T-S waves, developing a new control technique to transition from turbulence to laminar. Oscillating Surface (PDOS) actuators were used at three corresponding streamwise locations. These actuators act as dynamic surface modification, which both introduce artificial T-S waves, and subsequently cancel these waves. In the study, two upstream actuators were used to excite and phase-lock the T-S waves through systematic oscillation. The downstream actuator was used to cancel the T-S waves by applying an anti-phase disturbance at the proper amplitude. Particle Image Velocimetry was used to observe the development of the T-S waves and to characterize the three-dimensional development of the T-S waves along the airfoil model. The results indicate a dampening of the T-S wave magnitude, resulting in a delay of the transition of laminar to turbulent flow in the presence of a known adverse pressure gradient.
8:11AM Q33.00003 Nonlinear Performance of Linear Sensor-Based Output Feedback Control of Transitional Channel Flow

YIYANG SUN, HUAIJIN YAO, MAZIAR S. HEMATI, University of Minnesota

A large transient energy growth (TEG) of small flow perturbations can lead to laminar-to-turbulent transition in channel flow at sub-critical Reynolds number. Full-state feedback control, such as linear quadratic regulation (LQR), has been demonstrated to suppress TEG and prevent transition; however, access to full-state information is rarely possible in practice. In this study, we investigate two sensor-based output feedback control strategies: (1) static-output-feedback LQR and (2) linear quadratic Gaussian (LQG) control. These controllers use a few sensor measurements at the walls to determine the control input. We study the nonlinear effects on TEG behavior and transition control effectiveness by performing direct numerical simulations (DNS). We find that the nonlinearity serves to saturate TEG, with amplification being reduced as the initial disturbance amplitude is increased. Accordingly, the reduction of TEG in the controlled case also decreases, indicating a degradation of control performance when nonlinear effects become substantial. For LQG control, the nonlinear effects on the relation between physical and estimated states are examined. We investigate the estimator performance in DNS and further assess its influence on the control results.

8:24AM Q33.00004 Resolvent-based Modelling, Estimation and Control of the Cylinder Wake

BO JIN, RICHARD SANDBERG, SIMON ILLINGWORTH, The University of Melbourne

We use a resolvent based approach for the modeling, estimation and control of the cylinder wake at Reynolds numbers between 40 and 120. The work has three parts. First, we consider the optimal estimation problem in which a single sensor is used to estimate the whole flow field. Second, we consider full-information control in which a single actuator uses knowledge of the entire flow field for control. Third, we consider feedback control with a single sensor for measurement and a single actuator for control. A range of Reynolds numbers is considered and the trends of optimal sensor and actuator placements for estimation and control are presented.

8:37AM Q33.00005 Experimental Study of Laminar Separation Bubbles with Active Flow Control

DAVID BORGMAN, JESSE LITTLE, University of Arizona

Laminar separation bubbles (LSB) that form under the influence of an adverse pressure gradient are examined in a low turbulence wind tunnel. The LSBs are formed along the surface of a flat plate using a displacement body that is mounted from above. Rapid formation of spanwise vortical structures due to the primary shear layer instability promotes transition to turbulence and determines the extent of the separated region. Direct numerical simulations have shown the effectiveness of vortices that exploit the shear layer instability for active flow control (AFC). AFC in the form of 2D disturbance waves leads to the formation of coherent spanwise vortical structures that can reduce or completely eliminate the LSB. With a properly chosen forcing frequency and amplitude, the secondary absolute instability can be suppressed thus delaying transition and even relaminarizing the flow downstream of reattachment. The success of this AFC strategy is strongly affected by freestream turbulence where even modest levels can reduce the control authority due to excitation of Klebanoff modes. Therefore, the question arises whether the observed transition delay and relaminarization is possible in a real environment. This motivates a study of LSB AFC on a flat plate using DBD plasma actuators.

8:50AM Q33.00006 Suppressing flow separation over a flat plate using machine learning

AMIRKHOSRO KAZEMI, Florida Atlantic University

We examine supersonic flows over a 3D cavity at Mach number of 1.4 with cavity-depth-based Reynolds number of 10,000 using LES and propose a resolvent analysis based flow control technique to suppress the pressure fluctuations over the cavity. The relevant analysis identifies the range of the forcing frequency and spanwise wavenumber for the turbulent cavity flow where energy amplifies significantly. Given these insights from resolvent analysis, we perform a series of controlled flow simulations by prescribing the unsteady actuation with various combinations of the spanwise wavelength and actuation frequency over the supersonic turbulent cavity flow. The controlled achieves over 28% of reduction in pressure fluctuations. By combining the insights from resolvent analysis with DMD of the 3D cavity flows, we uncover two main mechanisms responsible for effective suppression: (1) the unsteady actuation thickens the shear layer near the leading edge and modifies its spreading rate to reduce the receptivity to acoustic disturbances; (2) the actuation disrupts the formation of large-scale spanwise vortices which mitigates the obstruction of incoming flow and alleviates the trailing-edge impingement.

9:03AM Q33.00007 Supersonic cavity flow control using resolvent analysis

QIONG LIU, University of California, Los Angeles

We examine supersonic flows over a 3D cavity at Mach number of 1.4 with cavity-depth-based Reynolds number of 10,000 using LES and propose a resolvent analysis based flow control technique to suppress the pressure fluctuations over the cavity. The resolvent analysis identifies the range of the forcing frequency and spanwise wavenumber for the turbulent cavity flow where energy amplifies significantly. Given these insights from resolvent analysis, we perform a series of controlled flow simulations by prescribing the unsteady actuation with various combinations of the spanwise wavelength and actuation frequency over the supersonic turbulent cavity flow. The controlled achieves over 28% of reduction in pressure fluctuations. By combining the insights from resolvent analysis with DMD of the 3D cavity flows, we uncover two main mechanisms responsible for effective suppression: (1) the unsteady actuation thickens the shear layer near the leading edge and modifies its spreading rate to reduce the receptivity to acoustic disturbances; (2) the actuation disrupts the formation of large-scale spanwise vortices which mitigates the obstruction of incoming flow and alleviates the trailing-edge impingement.


ANDRE POPINHAK, ROBERT MARTINUZZI, CHRIS MORTON, University of Calgary

Estimation of an unsteady velocity field with remote sensors is an important tool for many technological applications and an integral part of flow control strategies. Many works in the current literature have implemented sensor-based flow estimation via pressure-velocity correlations in wake-flows (e.g. Hosseini et al. 2015). Using the theoretical model of von Kármán Vortex Street, the aim of this work is to (i) understand whether sensor placement impact in the performance of the estimators. (ii) What is the benefit of using Quadratic Stochastic Estimation (QSE) relative to Linear Stochastic Estimation? (iii) What is the benefit of formulating QSE into a set of orthogonal regressors? It will be shown that the discrete pressure information location relaxes the training method requirement and a proper selection of an orthogonal basis for the estimator reduces overfitting.
Asymmetric instabilities in the flow of thin films on fibers. Chase Gabbard, Joshua Bostwick, Clemson University — Understanding the dynamics of thin-film flow down a vertical fiber is valuable for many coating applications. It is well known that such flows give rise to a number of instabilities that define the bead-on-fiber morphology. These include Plateau-Rayleigh breakup, isolated bead formation, and convective instabilities, all of which leave the interface shape axisymmetric. We conduct experiments that reveal an asymmetric instability in the flow of liquid on a fiber, which depends critically upon the liquid properties, flow rate and fiber radius, and we document this dependence on the observed morphology. Multiple liquids were used to change the viscosity and surfactant is used to control the surface tension in order to capture the transition between symmetric and asymmetric. Interestingly, we observe that under certain conditions in the transition region the instability may change from asymmetric to symmetric at some distance down the fiber.

Simulating instabilities of liquid metal alloys on nanoscale using molecular dynamics simulations.1 Ryan Allaire, New Jersey Institute of Technology, Miguel Fuertes-Cabrera, Oak Ridge National Laboratory, Philip D. Rack, The University of Tennessee, Knoxville, Linda Cummings, Lou Kondic, New Jersey Institute of Technology — Classical molecular dynamics simulations are used to investigate the influence of phase separation of liquid metal alloys on Rayleigh-Plateau (RP) type instabilities of free standing alloy lines, as well as alloys deposited on substrates. The alloy geometries are created in thin strips with widths modulated by sinusoidal waves of varying amplitudes and wavelengths, corresponding to the fastest growing mode obtained from continuum RP theory. We explore the influence of temperature on the breakup process and phase separation. Both the ratio of phase separation length scales to wavelengths of the sinusoidal perturbations, and the initial location of phase separation are found to influence the RP instability development, either by modifying the stability characteristics of material. The pattern formation process is robust: the crystal structure exhibits self-healing of initial or accidental defects. Existing theories (for example, spatio-temporal stability analysis) are adapted to our problem so as to rationalize our experimental results. In turn, we aim to take advantage of our model for the directed control of the instability toward the design of materials with prescribed properties and function.

Electrohydrodynamic patterning of non-Newtonian thin films. Adham RiaD, Hadi NazariPooR, Ali MohammadTabar, Mohtada Sadrzadeh, Department of Mechanical Engineering, University of Alberta — Electrically driven instabilities of thin liquid films or electrohydrodynamic (EHD) induced patterns has gained popularity for its ability to create nano-sized features for various applications. An initially quiescent thin film of uniform thickness is sandwiched between two electrodes. A voltage is then applied across the electrodes, creating a destabilizing electrostatic pressure on the interface, causing surface deformations and patterns to evolve. A mathematical model is developed to study the effect of power-law on the EHD patterning process. Simplifying the governing equations using the long-wave approximation leads to a novel thin-film equation that describes the interface’s dynamics for both pseudo-plastic and dilatant fluids. The spatiotemporal evolution of the patterning process is then numerically simulated by solving the nonlinear thin-film equation using a finite-difference based discretization scheme and an adaptive time step solver. The results show that the patterning time is strongly influenced by the power-law index and is significantly shorter for shear thickening than shear thinning fluids. Moreover, morphological changes between patterns of shear thinning and shear thickening fluids are correlated to local viscosity variations.

Electrohydrodynamic instability of viscous falling films: numerical verification of weakly nonlinear models and higher-order effects. Tao Wei, MengQi Zhang, National University of Singapore — A study on the nonlinear stability of an electrified film flowing under gravity down an inclined plate is performed with an electric field acting normal to the plate. An asymptotic expansion is applied to derive a nonlinear evolution equation for the interfacial position in the long-wave limit, which retains terms up to 2nd-order in the small parameter. Numerical solutions to two weakly nonlinear models, considering distinct effects of hydrostatic pressure, inertia and viscous dispersion, are compared to those of the fully nonlinear equation with an electric Weber number well above the critical value from linear analysis. Single- and multiple-kump solitary waves are searched for a weakly granular dispersion problem. Two coupled Ginzburg-Landau-Lindau equations are derived from a multiple-scale analysis, which suggests that the electric destabilization occurs in the form of travelling waves of finite amplitude. The leading-order solutions show that one order-of-magnitude increase in electric supercriticality will cause one order increase in the growth rate. Reasonable agreement is found between the present results and available experiments. In particular, for realistic parameter values an electric field can reduce the film thickness.

Thin water film rise on a rough surface of a vertical plate in water entry.1 Nayoung Kim, HyungMin Park, Seoul National University — A three-phase contact line is formed as a liquid film moves over the solid surface, and the maximum velocity at which the film does not separate from the surface is affected by several liquid properties, such as viscosity, density and surface tension. In addition, it has been shown that surface roughness can also encourage the early film moves over the solid surface, and the maximum velocity at which the film does not separate from the surface is affected by several liquid properties, such as viscosity, density and surface tension. The leading-order solutions show that one order-of-magnitude increase in electric supercriticality will cause one order increase in the growth rate. Moreover, morphological changes between patterns of shear thinning and shear thickening fluids are correlated to local viscosity variations.

1Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.
Simultaneous Liquid Flow and Drying on Rotating Cylinders

CHANCE PARRISH, SATISH KUMAR, Department of Chemical Engineering and Materials Science, University of Minnesota — The coating and drying of non-flat discrete objects is an important manufacturing step for a broad variety of products. Flow of a thin, non-volatile liquid film on the outside of a rotating cylinder is commonly used as a model problem for these processes. Here, we use lubrication theory to study the behavior of a volatile, particle-laden coating. Two coupled evolution equations describing variations in coating thickness and composition as a function of time and the angular coordinate are solved numerically. In the limit of a rapidly-rotating cylinder, results from linear stability analysis and nonlinear simulations demonstrate that non-uniform drying at larger drying rates may cause thickness and composition disturbances to regrow after initially decaying. When gravity is reincorporated, poor leveling of the coating thickness at lower rotation rates and larger drying rates leads to less uniform coatings. Colloidal particles hinder leveling at high concentrations through increases in the viscosity, but help prevent coating rupture at more moderate concentrations, leading to a composition “sweet spot”. A parametric study is then used to show that thickness and composition variations are minimized at a large rotation rate, low drying rate, and moderate particle concentration.

Three-dimensional numerical simulations of a thin film falling vertically down the inner surface of a rotating cylinder

JASON STAFFORD, University of Birmingham, UK, CAMILLE PETIT, OMAR MATAR, Imperial College London — A flow in which a thin film flows due to gravity on the surface of a rotating cylinder is investigated. This was performed using high resolution three-dimensional direct numerical simulations and a volume-of-fluid approach to treat the interface. The variation of the Ekman number (Ek), defined to be proportional to the rotation of the cylinder, and has a significant effect on various parameters of the flow. The centrifugal force increases with rotational speed, producing a stabilising effect (Iwasaki and Hasegawa, 1981), supressing wave formation. Key features, such as the transition from a 2D to a more complex 3D wave regime, and the local thickness of the film, are heavily influenced by this stabilisation and are investigated. Furthermore the imposed rotation results in distinct characteristics in the flow such as the development of angled waves due to the resolved velocity in the axial and azimuthal directions. Fast Fourier Transforms of the interface are performed which show how the wavelength and angle of the waves varies with Ek. Simulations have been conducted for a range of Ek to provide detailed insight on how this parameter affects the flow.

Thermally driven coalescence in thin liquid film flowing down a fiber

CLAUDIA FALCON, HANGJIE JI, Department of Mathematics, University of California, Los Angeles, ABOLFAZL SADEGHPOUR, ERFAN SEDIGHI, Y. SUNGTAEK JU, Mechanical and Aerospace Engineering Department, University of California, Los Angeles, ANDREA BERTOZZI, Department of Mathematics & Mechanical and Aerospace Engineering Department, University of California, Los Angeles — We aim at understanding the dynamics of thin fluid film flowing down a vertical fiber under streamline thermal effects, both experimentally and theoretically. Recent studies have shown the importance of determining the regime transition from absolute to convective instability. Unlike previous work, our experiments demonstrate that the onset of such irregular wavy regime can also be induced by thermal gradient away from the nozzle. The new model includes spatial-dependent viscosity and surface tension due to inhomogeneous temperature field along the fiber. The predicted coalescence positions based on this theory are useful in the design of heat and mass exchangers for applications that include cooling systems and desalination.

Rapid and multiplexed enrichment of specific DNA sequences using isotachophoresis

ASHVIN RAMACHANDRAN, Stanford University, NOBUYUKI FUTAI, Shibaura Institute of Technology, CATHERINE HOGAN, KANAGAVEE MURUGESAN, NIAZ BANAEI, JUAN G. SANTIAGO, Stanford University — We use on-chip isotachophoresis (ITP) to create electric-field-driven shock waves of ion concentration, which are formed at the interface between a high mobility leading electrolyte (LE) and a low mobility trailing electrolyte (TE). Ionic species with mobilities bracketed by these electrolyte species focus at the LE-to-TE interface. For trace sample concentrations, multiple species co-focus, pre-concentrate by 10,000x and react inside a single, order 10 um wide zone. We apply ITP to extract and purify DNA targets from complex biological samples and to immediately co-focus these with synthetic DNA/RNA probes. We complete in 30 min hybridization reactions which would normally take several days, and then separate reacted from unreacted DNA. We will present our work toward rapid and sequence-specific enrichment of rare DNA targets for two applications. First, we are developing miniaturized microfluidic capillary electrophoresis chips for use in Point-of-Care (PoC) devices, where our unique hybridization platform can perform miniaturized high-throughput DNA/RNA sequencing. Second, we have adapted our enrichment protocol for use in a clinical setting, where we can use our protocol to boost the sensitivity of a commercial PCR assay for tuberculosis. Here, we will present preliminary results from both applications.

Fast and homogenous mixing in a coaxial capillary device with two sheath flows

DIEGO A. HUYKE, ASHVIN RAMACHANDRAN, Stanford University, THOMAS KROLL, DANIEL P. DEPONTE, SLAC National Accelerator Lab, JUAN G. SANTIAGO, Stanford University — We have developed a novel microfluidic mixer with order 10 microsecond mixing times, and sample consumption of 1 to 500 nL/s. Importantly, the mixer achieves flow-area-weighted residence time distribution with a standard deviation width of 140 microsecond (for a 2.8 ms center-streamline residence time). In our mixer, the low flow rate sample stream which exits an inner capillary is hydrodynamically focused to a sub-micron radius by the high flow rate sheath stream within a tapered middle capillary. The mixed stream subsequently enters a third, outer capillary wherein its area (normal to the flow direction) is expanded 200 times to increase the sample detection volume. The latter expansion of the mixed stream decouples the upstream mixing region from the downstream probing region and increases the signal-to-noise (SNR) for line-of-sight integration techniques. The outermost capillary is constructed from glass or polyimide for, respectively, optical or hard X-ray sample detection. Analytical and numerical convection-diffusion models, design, and experimental validation of the mixer will be presented. The models will explore tradeoffs between mixing rate and homogeneity for different flow conditions. We validated the models and experimentally studied mixing performance using epifluorescence imaging of fluorescein-iodide quenching.
8:11AM Q35.00003 Joule heating-enabled nanoparticle enrichment in insulator-based dielectrophoretic microdevices. AMIRREZA MALEKANFARD, ZHIJIAN LIU, XIANGCHUN XUAN, Clemson University — Dielectrophoresis has become a popular method for particle manipulation in microfluidic devices. It requires electric field gradients that can be created using in-channel hurdles and posts etc. However, these insulating structures amplify the Joule heating effect, leading to a non-uniform fluid temperature and in turn gradients in fluid conductivity and permittivity (as well as viscosity). The action of electric field on the Joule heating-induced fluid property gradients causes a regional fluid flow in the form of vortices. This so-called electrothermal flow has been previously found to reduce the particle focusing and trapping performance in insulator-based dielectrophoretic microdevices. We demonstrate in this work that the electrothermal flow vortices can be utilized to entrain nanoparticle-based localized enrichment near the insulating tips of a ratchet microchannel. We also develop a depth-averaged numerical model to simulate the fluid, heat, charge and mass transport involved in the process. A good agreement is obtained for each of the various parametric studies.

8:24AM Q35.00004 Rapid mixing utilizing microscale magnets based on microfluidics device. RAN ZHOU, MARCEL MEIJULU, Purdue University Northwe...tualization of the nanowire diode. This non-contact measurement method suggests a relation between the permanent dipole and the depletion width of the nanowire diode. In this paper, we propose a miniaturized and integrated microfluidic device that can achieve the rapid mixing of ferrofluid and distilled water. To accomplish this, high-gradient microscale magnet was fabricated and integrated at one side of a microfluidic channel by a simple fabrication technique. The microscale magnet was fabricated by injecting and curing a mixture of neodymium powder in a structural channels, and subsequent permanent magnetization. This study further investigates the effect of the total flow rate and ferrofluid concentration on the mixing performance.

8:37AM Q35.00005 Electrical Double Layers: Predicting Overcharging and Layering of Ions using Continuum Models. ANKUR GUPTA, ANANTH GOVIND RAJAN, EMILY CARTER, HOWARD STONE, Princeton University — Electrical double layers (EDLs) form the basis for several phenomena such as electrophoresis, supercapacitor charging and discharging, and desalination. However, existing continuum models for the EDL are unable to predict experimentally-observed phenomena such as ion layering and EDL overcharging. To overcome these limitations, atomistic methods like classical density functional theory and molecular dynamics simulations have been employed to elucidate the nanoscale structure of EDLs. In this work, we bridge the gap between the continuum and atomistic approaches by proposing a modified continuum model. Our model predicts a near-wall stratified structure in EDLs wherein the cations and anions are arranged in layers, even in the dilute limit. Furthermore, we predict that for trivalent ions, the double layer can locally possess a net charge larger than the surface, a phenomenon commonly known as overcharging. Our model does not require any fitting parameters or additional boundary conditions, and only depends on the ion properties since our approach is at the continuum scale, we envision that it can be readily extended to out-of-equilibrium systems such as electrophoresis and dynamics of supercapacitors.

8:50AM Q35.00006 Electroosmotic flow in small-scale channels induced by surface-acoustic waves. MATHIAS DIETZEL, STEFFEN HARDT, Institute for Nano- and Microfluidics, TU Darmstadt — Apart from acoustic streaming as the primary effect, surface-acoustic waves (SAWs) go along with time-periodic changes of the electrostatic potential at a solid-liquid interface. This surface potential polarizes the liquid in its vicinity, i.e. it induces a dynamic Debye layer. Alternating current electroosmosis (ACEO) relies on a similar principle. We study the flow field due to the interaction of the dynamic Debye layer with the electric field of the SAW. For this purpose, we consider parallel-plates channels less than 500 nm wide, characterized by an inverse RC time of the Debye layer similar to the characteristic SAW frequencies in the MHz range. By means of numerical simulations of the unsteady Stokes, Poisson, and Navier-Stokes equations, parametric studies are conducted, varying the SAW frequency and amplitude, ionic strength, Debye parameter, as well as the phase shift between two SAWs traveling along opposite channel walls. We demonstrate that for typical SAW amplitudes sizable (time-averaged) net velocities up to 1 mm/s can be obtained, especially if the phase shift equals 180 deg to maximize the periodic electric current between the walls. This significantly exceeds the velocities induced by acoustic streaming in narrow channels.

9:03AM Q35.00007 Stern compact layer in ionic conductor liquid charging at high voltages1. FARZAD MASHAYEK, BABAK KASHIR, ANTHONY E. PERRI, ALEXANDER L. YARIN, University of Illinois at Chicago — The Stern compact layer at the conducting electrodes is studied theoretically and numerically. These electrodes are subjected to high voltages and sustain electric current. A novel approach is developed based on the Brunauer-Emmet-Teller (BET) mechanism to predict the thickness of the Stern compact layer. Non-specific (non-electric) adsorption is responsible for the formation of this layer on the oxide islands or impurities at the conducting electrodes. Concurrently, kinetics-limited faradaic reactions occur at the unscreened parts of the metallic conducting electrodes. The electron transfer occurs through the faradaic reactions characterized by the Frumkin-Butler-Volmer kinetics. The model relates the thickness of the Stern compact layer to the potential drop across it. Realistic values of the applied electrode voltage and sustained electric current density in electrostatic atomization are considered to predict the Stern compact layer properties.

9:16AM Q35.00008 A non-contact solution-based method to measure the electric dipoles of p-n silicon nanowire diodes. MINH THANG HOANG, Rutgers University, GOZDE TUTUNCUOGLU, AMAR MOHABIR, Georgia Institute of Technology, LEONARD FELDMAN, Rutgers University, MICHAEL FILLER, Georgia Institute of Technology, JERRY SHAN, Rutgers University — Dispersions of nanowires with p-n junctions are emerging as next-generation electronic colloidal materials. Developing the ability to efficiently manipulate such nanostructures for both characterization and separation based on electronic properties is crucial for scientific understanding and technological applications. Here we present a method for measuring the permanent and induced dipoles of p-n doped silicon nanowires by observing their rotation in fluid suspension under external electric fields. The permanent dipoles are formed by the opposite charges in the depletion layer at the p-n junction, while the induced dipoles are formed by nanowire polarization under the external field. The measured electrical torque exerted on the p-n nanowires due to applied DC and AC fields yields the permanent and induced dipoles. We then show how the dipoles change for nanowires of various doping concentrations and surface passivations. Numerical results also suggest a relation between the permanent dipole and the depletion width of the nanowire diode. This non-contact measurement method offers a new approach to efficiently determine the electronic properties of p-n nanowires, and is a significant step toward the high-throughput characterization and separation of complete nanowire devices.

1This work was supported by the US National Science Foundation grant CBET-1505276.
Comparing to numerical simulations, we show that the mixing time can be predicted by using the same of diffusiophoresis in the mixing process through an effective Péclet number built on this modified Batchelor scale. Whilst this small scale is

system. We show that the microfluidic chips reproducibly generate droplets with frequencies in the order of 10kHz and droplet sizes between

work we study the interaction of liquid and gas in flow-focusing microchannels with the goal of reproducible generation of microscale droplets

will provide a means to establish the fundamental understanding required to estimate apparent deformability of flowing objects in high-speed.
inertially-focus deformable droplets in Poiseuille flow. The results suggest that accurate measurements of lateral positions of deformable droplets

lateral equilibrium position were determined using high-speed microscopy. Through dimensional analysis, we identified critical conditions to

Drops, MARINA EFSTRATIOU, JOHN CHRISTY, KHELLIL SEFIANE, Institute of Multiscale Thermofluids, School of Engineering,

interfacial tension between droplets and continuous phase. This work shows the possibility to process high-viscosity fluid droplets in a variety

in methanol or ethanol to produce regular patterns that are injected into a focusing section with another alcohol solvent. To probe the role

the continuous phase is experimentally investigated in microchannels. Microfluidic segmented flows of oil droplets are continuously generated

tension is driven by concentration gradients occurring due to crystal nucleation, demonstrating for the first time how crystal nucleation affects the flow.

III is governed by the appearance of strong flow jets, directed at the growing crystal, and vortices to either side of the crystal. This flow regime

ring on the periphery of the drops. Three evaporation stages were observed. During stage I, a generally outward flow is manifested driven by

for the first time the existence of a nucleation-driven flow within the droplets which appears to be responsible for the formation of the crystal

micro-PIV (Particle Image Velocimetry) to examine the link between the flow along the base of the drop and the final deposit. We report

on the sign of the diffusiophoretic coefficient $D_{diff}$. Mixing is delayed when $D_{diff} > 0$ (salt-attracting configuration), or faster when

$D_{diff} < 0$ (salt-repelling configuration). In both configurations, as for molecular diffusion alone, large scales are barely affected in the dilating
direction while the Batchelor scale for the colloids, $\ell_{c, diff}$, is strongly modified by diffusiophoresis. We propose here to measure a global effect of diffusiophoresis in the mixing process through an effective Péclet number built on this modified Batchelor scale. Whilst this small scale is obtained analytically for the stagnation point, in the case of chaotic advection, we derive it using the equation of gradients of concentration, following Raynal & Gence (1997). Comparing to numerical simulations, we show that the mixing time can be predicted by using the same function as in absence of salt, but as a function of the effective Péclet numbers computed for each configuration.

1 ANR tunamix, breakthrough IDEX Lyon Turbulent

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q36 Microscale Flows: Drops and Bubbles 618 - Prashant Valluri, University of Edinburgh

7:45AM Q36.00001 Deformability-induced inertial focusing of viscous droplets1, SOO-JUNG HUR, MEHRAN ABOULGHASEMIBIZAKI, Johns Hopkins University — Deformability-induced lateral migration of soft objects across the streamlines in microchannels provides a robust and continuous way of manipulating droplets and cells inflow. Theoretical analysis has shown that there exists a force away from channel walls in Poiseuille flow that locates deformable particles closer to the channel center than rigid counterparts. As lateral lift forces acting on flowing objects scale with carrier fluids’ properties, the deformability can be extrapolated from the positions of objects with known sizes in the channel. Here, behaviors of nearly-neutrally buoyant microscale droplets of various sizes, interfacial tensions, and viscosities were tested to determine droplet’s physical attributes and flow conditions, enhancing their lateral migration. Fluorinated oil solutions ($\mu =1.7\text{mPas}$ and $5\text{mPas}$) containing droplets ($1\text{mPas}<\mu<1.3\text{Pas}$) were injected into a microfluidic channel at $R_e<50$. The interfacial tensions were varied by adding a controlled amount of a surfactant. The diameter, deformability, and lateral equilibrium position were determined using high-speed microscopy. Through dimensional analysis, we identified critical conditions to inertially-focus deformable droplets in Poiseuille flow. The results suggest that accurate measurements of lateral positions of deformable droplets will provide a means to establish the fundamental understanding required to estimate apparent deformability of flowing objects in high-speed.

1 NSF CBET-1804004

7:58AM Q36.00002 Dynamics of High-speed Droplet Generation in Gas-liquid Microfluidic Systems1, POOYAN TIRANDAZI, JULIAN D. ARROYO, CARLOS H. HIDROVO, Northeastern University — In this work we study the interaction of liquid and gas in flow-focusing microchannels with the goal of reproducible generation of microscale droplets in air. The microfluidic channels are fabricated in Polydimethylsiloxane (PDMS) based on established lithography techniques and feature a non-planar architecture to enable the formation of liquid jets and droplets within air. A comprehensive flow map is developed based on the interaction of liquid and gas for a wide range of flow conditions and different channel sizes. In particular, we focus on the characteristics of the Dripping and Jetting modes of droplet formation and present information regarding droplet and jet sizes and breakup frequencies in this system. We show that the microfluidic chips reproducibly generate droplets with frequencies in the order of 10kHz and droplet sizes between $160\mu$m and $50\mu$m in the Dripping mode, whereas, for the Jetting we obtain droplets from $50\mu$m down to $15\mu$m at frequencies higher than 100 kHz. Finally, we briefly discuss some applications of high-speed gas-liquid microfluidics, namely for oil-free polymer particle fabrication and gas sensing using sample digitization with microdroplets.

1 We appreciate the financial support from the National Science Foundation (NSF award no. 1805244).

8:11AM Q36.00003 Microfluidic droplet manipulations via modulation of interfacial tension1, THOMAS CUBAUD, Stony Brook University — The dynamic response of viscous droplets to variations of interfacial tension with the continuous phase is experimentally investigated in microchannels. Microfluidic segmented flows of oil droplets are continuously generated in methanol or ethanol to produce regular patterns that are injected into a focusing section with another alcohol solvent. To probe the role of fluid properties on droplet deformations, a range of fluid combinations are examined with focus on large droplet elongations for vanishing interfacial tension between droplets and continuous phase. This work shows the possibility to process high-viscosity fluid droplets in a variety of solvents in confined microsystems.

1 This work is supported by NSF (CBET-1150389).

8:24AM Q36.00004 Crystallisation-Induced Flows in Evaporating Aqueous Saline Drops, MARINA EFSTRATIOU, JOHN CHRISTY, KHELLIL SEFIANE, Institute of Multiscale Thermofluids, School of Engineering, University of Edinburgh — When aqueous saline droplets are left to dry on hydrophilic glass slides, a ring of spaced crystals is formed. However, no experimental work has yet linked the flows arising in these droplets with the final crystal ring deposition. In our work, we have performed micro-PIV (Particle Image Velocimetry) to examine the link between the flow along the base of the drop and the final deposit. We report for the first time the existence of a nucleation-driven flow within the droplets which appears to be responsible for the formation of the crystal ring on the periphery of the drops. Three evaporation stages were observed. During stage I, a generally outward flow is manifested driven by evaporative flux, since the evaporation rate is higher on the periphery. After stage I, a transition stage II is shown during which the flow almost momentarily pauses, that is believed to occur at the point where the concentration near the periphery is that of the incipient nucleation. Stage III is governed by the appearance of strong flow jets, directed at the growing crystal, and vortices to either side of the crystal. This flow regime is driven by concentration gradients occurring due to crystal nucleation, demonstrating for the first time how crystal nucleation affects the flow.
In this work, we study the emergence of superwalking behavior in droplets. Superwalkers are droplets that travel at speeds greater than triple the speed of the fastest ones, and enable a plethora of novel multi-droplet behaviors. Physical insights from numerical simulations confirm that superwalkers arise from the interplay between the droplet and its wave field. We have studied walking droplets in the presence of two driving frequencies and have observed a new type of droplet breakup, where two aqueous phases protrude simultaneously from the branches, leading to the simultaneous droplet breakup from two interfaces. This phenomenon suggests that the changes in pressure between the interfaces and within the droplets play an important role in the emergence of superwalking behavior. Our experimental results are validated against the results obtained from this analytical model.

This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (2017R1A6A3A04006179) and the 2019 Research Fund (1.190122.01) of UNIST (Ulsan National Institute of Science and Technology).
7:58AM Q37.00002 Experimental and theoretical investigation of capillary oscillations of a charged droplet levitated using various applied waveforms. NEHA GAWANDE, MOHIT SINGH, Y.S. MAYYA, ROCHISH THAOKAR, Indian Institute of Technology Bombay, Mumbai — A conducting charged droplet levitated in a quadrupole trap becomes unstable when the surface charge on the droplet exceeds its Rayleigh limit. However, when the charge on the droplet is in the sub-Rayleigh limit, it exhibits capillary oscillations whose amplitude and frequency depend upon the magnitude of the charge as well as on the applied electric field. In this work, we present the high-speed imaging of the surface oscillations of a highly charged droplet levitated using various applied waveforms such as sine, square and ramp wave. The surface dynamics is characterized by Fast Fourier Transformation (FFT) analysis and it is observed that the droplet oscillates with the applied frequency irrespective of the type of the waveform. To understand the experimental observation an asymptotic theory is carried out for a perfectly conducting charged droplet in the potential flow limit with viscous corrections. The analysis suggests that the droplet oscillation behavior is the result of complex interplay between the dipolar charge (FFT) analysis and it is observed that the droplet oscillates with the applied frequency irrespective of the type of the waveform. To understand the experimental observation an asymptotic theory is carried out for a perfectly conducting charged droplet in the potential flow limit with viscous corrections. The analysis suggests that the droplet oscillation behavior is the result of complex interplay between the dipolar charge.

8:11AM Q37.00003 Rheological measurements of gels via ultrasonic levitation of gel drops1. J.R. SAYLOR, XINGCHEN SHAO, Clemson University, STEVEN FREDERICKS, University of Minnesota, JOSHUA BOSTWICK, Clemson University — The application of ultrasonic levitation to the measurement of surface tension of liquid drops has a rich history. However this method has not been extended to gels which, unlike liquids, have a finite elasticity. Herein a method is presented for obtaining measurements of elasticity, surface tension, and viscosity of ultrasonically levitated gel drops. Agarose, a hydrogel, was the material explored. This approach is a significant development given that gels are of growing importance due to their relevance to biomedical applications and exhibit behaviors partially determined by their elasticities. Moreover, obtaining surface tension of gels is important but challenging since measurements cannot be made using standard Wilhelmy plate or DuNuoy ring methods, each of which cannot be applied without breaking the gel. Herein a theoretical development is presented which enables obtaining elasticity, surface tension, and viscosity of a gel drop from characteristics of its response to ultrasonic excitation. Measurements of surface tension and viscosity obtained using this approach are obtained for gel drops having viscosities ranging from 12.2 Pa to 200.3 Pa.

1Support from NSF Grant CBET-1750208 is acknowledged.

8:24AM Q37.00004 Delayed coalescence of drops at moving liquid/liquid interfaces2. WEHELIYE HASHI WEHELIYE, TENG DONG, ThAMeS Multiphase, Department of Chemical Engineering, University College London, WCIE 6BT, FEI WANG, Beijing Institute of Space Long March Vehicle, No1. Nadandhongmen Road, Fengtai District, Beijing, 100076, China, PANAGIOTA ANGELO2, ThAMeS Multiphase, Department of Chemical Engineering, University College London, WCIE 6BT — In this work, the delayed coalescence of drops with moving interfaces between two immiscible liquids were experimentally investigated. The aqueous phase was 78% Glycerol/water solution and the organic phase was Exxsol D80 oil. It was found that the coalescence was delayed when the interface moved and the delay time increased with the speed of the interface. The delay was attributed to the lubrication pressure in the film trapped between the drop surface and the bulk interface. Particle Image Velocimetry (PIV) was used to study the velocity fields while the structure of the thin trapped film was investigated with the Planar Laser Induce Fluorescence (PLIF) technique. The film was found to form a dimple symmetrical to the centerline when the interfaces had low velocity, while the dimple was less obvious when the interface velocity increased. Numerical simulations were carried out to investigate the profile of the lubrication pressure along the film of varying thickness.

2Corresponding Author

8:37AM Q37.00005 Interfacial Instability in Spheres by Resonance, NEVIN BROSIOUS, KEVIN WARD, University of Florida, TAKEHIKO ISHIKAWA, SATOSHI MATSUMOTO, Japan Aerospace Exploration Agency, MIKE SANSOUCIE, NASA Marshall Space Flight Center, RANGA NARAYANAN, University of Florida, UNIVERSITY OF FLORIDA TEAM, JAPAN AEROSPACE EXPLORATION AGENCY COLLABORATION, NASA MARSHALL SPACE FLIGHT CENTER COLLABORATION — When a levitated spherical liquid drop is subjected to continuous periodic forcing at a frequency equal to one of its natural frequencies, it can undergo resonance and form modal structures at the surface. The natural frequencies of a liquid sphere directly depend on the modal structure, the mass of the liquid, and the surface tension. By deliberately resonating a sphere at its natural frequency we can therefore obtain the surface tension. The work presented herein compares the analytical result for natural frequency of a liquid sphere in a “self-gravitational field” by Rayleigh (1879) to experimental observations using levitated water in ambient conditions and molten metals for varying modal structures. The natural frequency of two normal modes are obtained to verify the values of surface tension. Comparisons and contrasts between experiments and theory are explained. A method for the measurement of interfacial tension of high temperature liquid metals is introduced. Acknowledgments: NASA 80NSSC18K1173, NASA NNX17AL27G, FSGC08/NNX15025, UFIC Research Abroad for Doctoral Students Award

8:50AM Q37.00006 Levitation of a non–volatile drop by an evaporating pool: the inverse Leidenfrost effect, S.J.S. MORRIS, UC Berkeley, MENG SHI, KAUST — Assuming axisymmetry, zero gravity, and uniform surface tension $\gamma$ and vapour pressures (viscosity $\eta$, conductivity $k$ and density $\rho$), we determine the maximum value of the force $F$ with which a heated sphere (radius $b$) can be pressed against the pool surface without rupturing it. The Laplace–Young and Reynolds equations form a coupled system of ODEs determining, in particular, film thickness $h_0/b$ at the sphere bottom as a function of $F/(2\pi b)$ with $\varepsilon = \frac{\pi k T}{\rho b^2} \Delta T$, as a parameter (latent heat $H_0$). Numerical solutions for fixed small $\varepsilon$ show that as $F/(2\pi b)$ is increased from zero, $h_0/b$ first decreases to a minimum. With further increase in $F$, $h_0$ increases until a turning point is reached. There, the slope $\partial h_0/\partial F \to \infty$, and the response curve doubles back on itself to form an upper branch. Near the turning point, the interface shows an apparent contact line with apparent contact angle $\pi$ (on the liquid side). The turning point corresponds to the contact line moving from the lower hemisphere to the upper; during this process, $F/(2\pi b)$ reaches its maximum (unity). This result is consistent with work by Adda–Bedia et al.(2016).
The authors gratefully acknowledge support from the following sources: P.S. from the NSF, RTG/DMS-1646339. Z.C., S.Y.L from the SURE offered by Courant Institute of Mathematical Sciences, New York University.

9:29AM Q37.00009 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q38 Porous Media Flow General III 620 - Pejman Sanaei, NYIT

7:45AM Q38.00001 Effects of particles diffusion and membrane pore elasticity on membrane filtration performance1, PEJMAN SANAEI, New York Institute of Technology, SHI YUE LIU, ZHENGYI CHEN, New York University — Membrane filters fouling, which is an inevitable consequence of particle removal from the feed solution, is sensitive to the flow rate and the internal morphology and structure of membrane. In a very slow filtration process or during the late stage of filtration, when the flow is naturally very slow and Peclet number is small, particle diffusion is essential and cannot be neglected. Beside this, real membranes have complex geometry, and consist of a series of bifurcating elastic pores, which decrease in size as the membrane is traversed. In this talk, we introduce two first-principle models considering asymptotic analysis based on the membrane pores aspect ratio and a distinguished limit of the particle Peclet number. We consider the effects of diffusion for a single membrane pore as well as elasticity in a membrane with branching structure on filtration performance. In the first model, pressure driven flow is considered through the pore and advection-diffusion equation for the particle concentration is coupled with novel fouling models. Furthermore, in the second model, we investigate the membrane pores evolution under two different forcing mechanisms (constant pressure and flux) and describe how the membrane is traversed. In this talk, we introduce two first-principle models considering asymptotic analysis based on the membrane pores aspect ratio and a distinguished limit of the particle Peclet number. We consider the effects of diffusion for a single membrane pore as well as elasticity in a membrane with branching structure on filtration performance. In the first model, pressure driven flow is considered through the pore and advection-diffusion equation for the particle concentration is coupled with novel fouling models. Furthermore, in the second model, we investigate the membrane pores evolution under two different forcing mechanisms (constant pressure and flux) and describe how the membrane internal morphology changes due to its fouling and elasticity.

1The authors gratefully acknowledge support from the following sources: P.S. from the NSF, RTG/DMS-1646339. Z.C., S.Y.L from the SURE offered by Courant Institute of Mathematical Sciences, New York University.

7:58AM Q38.00002 Simulations of small particle deposition on a membrane filter pore using the immersed boundary method, SCOTT WEADY, New York University - Courant Institute, PEJMAN SANAEI, New York Institute of Technology — Membrane filters have widespread use in the medical, biotech and food science industries, among many others, and their design relies on modeling the flow and fouling mechanisms relevant to each specific application. In this talk, we describe immersed boundary method simulations of the fouling of a membrane filter pore by small particle deposition. The distribution of particles evolves according to an advection-diffusion equation driven by Stokes flow which is coupled to an adhesion model. We consider pressure driven flow through a rigid pore as well as constant flux through an elastic pore, and compare our results with those of several asymptotic models based on the order of the Péclet number and the pore aspect ratio.

8:11AM Q38.00003 Asymmetric Two-phase Flows Resistance in Homogeneous and Heterogeneous Anisotropic Porous Microstructure1, DARIO MAGGIOLO, Chalmers University of Technology, FEDERICO TOSCHI, Eindhoven University of Technology, FRANCESCO PICANO, University of Padova, SRDJAN SASIC, HENRIK STRÖM, Chalmers University of Technology — Two-phase flows in porous media exhibit anomalous behaviours at low capillary numbers due to the complex mechanism of interaction between flow spatial configuration and topology of the microstructure. In this study, we investigate the asymptotical nature of the two-phase flow resistance induced by the anisotropic features of the porous microstructure. We perform pore-scale direct numerical simulations of two-phase flows in porous media composed of solid particles with different shapes and orientations, using the Lattice-Boltzmann method. The results indicate that the infiltration of a fluid into a single pore is regulated by the topological traits of the pore, including its anisotropy. These traits determine a geometrical characteristic length of the pore \( \ell_p \), quantifying the flow resistance, which is directional-dependent: If the capillary length \( \ell_c = \gamma / \rho \) (i.e., the ratio between surface tension and capillary pressure) falls below the characteristic pore length \( \ell_c < \ell_p \), pore infiltration occurs, otherwise the fluid remains trapped. We extend the analysis to heterogeneous anisotropic microstructure in order to investigate the effect of the spatial configuration of the pores on the global flow resistance.

1This project has received funding from the European Unions Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No 790744.
8:24AM Q38.00004 Hydrodynamic driven dissolution in porous media with fluid-filled cavities, MOJDEH RASOULZADEH, WYATT KUEHSTER, University of Alabama — Hydrodynamics is a key player in defining the dissolution rate and dissolution hotspots in porous media with complex pore structure including embedded cavities. The cavity boundary, the fluid velocity maintains the concentration gradient and provides a fresh source of the solvent that facilitates dissolution. Given the characteristics of cavity and the porous zone, vorticities may form, and the cavity may partially or fully participate in the overall flow. In order to predict the dissolution hotspots properly, it is crucial to define the flow field accurately. We use the analytical models of flow in a porous medium including a random distribution of fluid-filled cavities. Darcy’s law is coupled to the Stokes flow for spherical shaped cavities. On the cavity boundary, a no-jump condition on normal velocities, jump on pressures, and the generalized Beavers-Joseph-Saffman condition on tangential velocities is applied. A sequential non-iterative approach is applied to handle the coupling between the hydrodynamics and dissolution. Transport of solute is facilitated by the spontaneous dissolution of solute at every grid point where the dissolved minerals is predicted by PHREEQC. Dissolution hotspots are detected.

8:37AM Q38.00005 Viscous Transport in Erodong Porous Media, SHANG-HUAN CHIU, BRYAN QUAIFE, NICHOLAS MOORE, Florida State University — Erosion is a fluid-mechanical process that is present in many geological phenomena such as groundwater flow. We present a boundary integral equation formulation to simulate two-dimensional erosion in porous media. One numerical challenge is accurately resolving the interactions between nearly touching eroding bodies at low porosity. We present a Barycentric quadrature method to resolve these interactions and compare it with the standard trapezoid rule. We compute the velocity, vorticity, and tracer trajectories in the geometries that include dense packings of 20, 50, and 100 eroding bodies. Like in our previous work [1], we observe quick expanding channels between close bodies, flat faces developing along the regions of near contact, and bodies eventually vanishing. Finally, having computed tracer trajectories, we characterize the transport inside of eroding geometries by computing and analyzing the tortuosity and anomalous diffusion rates. 1 B. D. Quaife et al., J. Comput. Phys. 375, 1 (2018).

8:50AM Q38.00006 ABSTRACT WITHDRAWN —

9:03AM Q38.00007 Experimental study of permeability of oriented fiber arrays, QIANGHONG WU, ZENGHAO ZHU, Villanova University — In this paper, a systematic study is performed to examine the permeability of a highly organized, oriented porous layer. Despite of extensive theoretical studies for the Darcy permeability of dilute or concentrated, oriented fiber array when the flow is either perpendicular or parallel to the fiber axis, there is a lack of research for the porosity between 0.3 and 0.8. Furthermore, no experimentally validated solutions have been reported to estimate the permeability of oriented fiber arrays where the angle between the fiber axis and the flow direction is in the range of 0 degree to 90 degree. We present in this paper an experimental study to examine the Darcy permeability of 3-D printed fiber arrays with different orientations. New correlations have been obtained when the porosity of the fiber array is in the range of 0.3 and 0.8. Furthermore, we have proved that, it is appropriate to use a linear combination method, based on the permeabilities of the fiber array at two distinct orientations, to predict the permeability of the fiber array at other orientations. The study presented herein has important applications in both biological systems and industrial applications, e.g. soft porous lubrication.

9:16AM Q38.00008 Dewatering saturated, networked suspensions with a screw press, TOM EAVES, DANIEL PATERSON, University of British Columbia, DUNCAN HEWITT, University of Cambridge, NEIL BALMFORTH, MARK MARTINEZ, University of British Columbia — A model is presented for the dewatering of a saturated two-phase porous medium in a screw press. The model accounts for the detailed two-phase rheological behaviour of the pressed material and splits the press into two zones, an initial well-mixed constant-pressure region followed by an axial transport region in which the total pressure steadily increases. In this latter region, a slowly-varying helical coordinate transformation reduces the dynamics to an annular bi-axial compression of the two-phase porous medium. Unlike previous modeling, the transition point between the two zones is determined self consistently, rather than set a priori, and the pressure along the length of the press is deduced from the rheology of the two-phase flow rather than averaging the two-phase dynamics over a cross-section of the press. The model is compared to experimental observations of the dewatering of a paper-making fibre suspension and of a clay slurry, and is shown to reproduce operational data.

9:29AM Q38.00009 Rayleigh-Taylor mixing in a porous medium, GUIDO BOFFETTA, MATTEO BORGNINO, STEFANO MUSACCHIO, University of Torino — Rayleigh-Taylor mixing in a porous medium is studied by high-resolution direct numerical simulations of the Darcy-Boussinesq equations in two and three dimensions. The width of the mixing layer is found to grow linearly in the limit of small diffusivity, in agreement with the dimensional expectation. A different growth rate is observed in two and three dimensions. The characteristic transverse scale, a measure of the typical plume size, grows slower following a diffusive law: as a consequence plumes became more elongated during the time evolution. The evolution of the density flux, quantified by the Nusselt number, is studied as a function of the Darcy-Rayleigh number.

1 National Science Foundation, Award No. 1511096.

1 INFN FieldTurb initiative and CINECA "INF19 fldturb"

Tuesday, November 26, 2019 7:45AM - 9:42AM –
Session Q39 Geophysical Fluid Dynamics Stratified Flow III 6a - Alexis Kaminski, University of Washington

7:45AM Q39.00001 3D measurements of inclined vortex rings interacting with a density stratification, JOHAN PINAUD, JULIE ALBAGNAC, PIERRE BRANCHER, SEBASTIEN CAZIN, ZEINAB RIDA, Institut de Mecanique des Fluides de Toulouse — Vortex rings are coherent vortical structures that dominate the dynamics of numerous flows as they are generated each time an impulsive jet occurs in a homogeneous fluid. They are also considered as elementary bricks of turbulence. Their faculty to propagate along their revolution axis by self-induction confers to such structures interesting transport properties, namely, transport of momentum, mass and heat. They are therefore often qualified as good candidates for mixing. From this perspective, the present study addresses the interaction of a vortex ring with a density stratification in order to get a better understanding of the subsequent mixing mechanisms. A new 3D time-resolved technique is used and gives a highlight at short timescale on the 3D vorticity reorganization and at larger timescale on the 3D patterns of internal gravity waves forced by the impacting/penetrating vortex. The influence of the Reynolds number of the vortex ring and its angle of attack relative to isopycnals will be detailed.
Dynamics of a reactive spherical particle falling in a linearly stratified layer

**Ludovic Huguet, Victor Barge-Zwick, Michael Le Bars, CNRS, Aix Marseille Univ, Centrale Marseille, IRPHE, Marseille** — The behavior of a particle falling in a stratified layer has already been studied for regimes of small Reynolds $Re$ or Froude $Fr$ numbers. However, the dynamics of a reactive particle have been unexplored, especially for regimes of interest for geophysical applications (large $Re$ and $Fr$ numbers). In a large water tank with linear stratification, reactive spheres made of a mixture of ice and hydrohalite solidifying below $-21.1^\circ \text{C}$, are released from the top and melt while they sink. PIV is used to track their falls and the dynamics of the surrounding environment. Results are compared with non-reactive plastic spheres. For large Reynolds and Froude number, the added drag (compared to a sphere in a homogeneous fluid) of the plastic spheres due to the stratification is proportional to $(Re/Re)_0.5$. For the reactive spheres, the added drag is much larger, suggesting a strong modification of the wake due to the melting. We also characterize the generation of internal waves and the associated radiated energy. While increasing with radius for plastic spheres, the ratio of wave energy compared to the initial potential energy of the spheres is constant over the explored range for reactive ones.

Grant Agreement No.681835FLUDYCOERC-2015-CoG

Analysis of one-dimensional models for exchange flows under strong stratification

**Herman Clercx, Steven Kapteyn, Eindhoven University of Technology, Vincenzo Armenio, University of Trieste, Matias Duran Matute, Eindhoven University of Technology** — One-dimensional models of exchange flows driven by horizontal density gradients are well known for performing poorly in situations with weak turbulent mixing. The main issue with these models is that the horizontal density gradient is usually imposed as a constant, leading to non-physically high stratification known as runaway-stratification. Here, we propose two new parametrizations of the horizontal density gradient leading to one-dimensional models able to tackle strongly stratified exchange flows at high and low Schmidt number values. The models are extensively tested against results from laminar two-dimensional simulations and are shown to outperform the models using the classical constant parametrization for the horizontal density gradients.

This research was funded by STW now NWO/TTW (the Netherlands) through the project "Sustainable engineering of the Rhine region of freshwater influence" (12682)

Flavours of Stratified Shear Flows: Algorithmic Detection

**Hesam Salehipour, Woods Hole Oceanographic Institution, Tom Eaves, University of British Columbia** — It remains unknown how to relate our understanding of turbulent mixing in stratified shear flows (based on direct numerical simulation (DNS)) to oceanic measurements of 1D profiles of density and velocity. In particular, there are various pathways to turbulence in stratified shear flows which behave in categorically different ways, but a method of distinguishing between these pathways in 1D microstructure measurements has yet to be proposed. Recently, a coordinate-free algorithmic classification scheme for the nonlinear states which result from the saturation of each instability has been proposed by Eaves and Balmforth (JFM 860, 2019), and hence is ideally suited for examining 1D profiles. This study was mostly restricted to relatively idealised steady, 2D flows; however, some aspects of this scheme were seen to carry over to non-steady, 3D flows after coarse-graining of the flow profiles. In this talk, we will investigate when the classification scheme works for realistic flows. In particular, we shall examine a large number of 1D profiles obtained from DNS of both Kelvin–Helmholtz and Holmboe instabilities to investigate the performance of this scheme over a large range of Reynolds and Richardson numbers, in addition to the type of instability and the 'age of the flow.'

Turbulent Mixing in Stratified Shear Flows

**Eric D'Asaro, Applied Physics Laboratory, University of Washington** — Turbulent mixing plays an important role in setting the distribution of heat, salt, and other biogeochemical tracers in the ocean, and quantifying the turbulent fluxes of these tracers is therefore a key question. A common approach is to look for signatures of turbulent overturns in one-dimensional profiles of temperature or other tracers and subsequently infer details of the accompanying fluxes and mixing. Often, these observed profiles are interpreted in the context of classical Kelvin-Helmholtz instability, in which shear drives the formation of a large overturn that subsequently triggers transition to turbulence. However, both the linear shear instability and nonlinear flow evolution can depend sensitively on the details of the background shear and stratification. Here we present the results of a series of direct numerical simulations of stratified shear instabilities with symmetric and asymmetric initial conditions, i.e. with either coincident or vertically offset profiles of shear and stratification. Motivated by recent Lagrangian float observations of the ocean transition layer, we examine the vertical structure of the resulting overturns. We describe both the size and stratification of these structures, and discuss implications for the associated buoyancy flux.

This work is supported by National Science Foundation grant OCE1657676.

Wake identification of stratified flows using Dynamic Mode Decomposition

**Chan-Ye Ohh, Geoffrey Spezding, University of Southern California** — Early experiments suggest that early wake information including body geometry and initial conditions in a linearly stratified fluid is lost during the wake evolution process (Meunier and Spezding Phys. Fluids 16, 296-305, 2004). Though this result was established for certain statistical quantities, it is less clear that there is no pattern remaining, and the process by which it is lost is also not established. Here we investigate how to identify, in principle, the various known regimes of stratified flow using a more sophisticated method, Dynamic Mode Decomposition (DMD), on 3D wake data from tomographic PIV and second-order DNS at low $Re (200 \leq Re \leq 1000)$ and low $Fr (0.5 \leq Fr \leq 8)$. Within the large set of modes, the dominant dynamic modes can be ranked and categorized into known regimes from a mode selection algorithm. The identification process is further refined and tested for spatially and temporally limited wake measurements.

Support from ONR Grant N00014-15-1-2506 under Dr. Peter Chang is most gratefully acknowledged.
9:03AM Q39.00007 Critical Reflection Dissipates Internal Wave Energy1, BRUCE RODENBORN, CLAYTON BELL, CHARLOTTE MABBS, Centre College — Ocean measurements show that continental slopes are eroded to the critical angle of local internal waves (Cacchione et al., Science 296, 2002). However, the feedback mechanism modifying the slopes over geologic time is not clearly understood. Other work shows that tidally driven topography creates strong boundary flows but weak tidal conversion (Dettner et al, Phys., Fluids, 25, 2013). We find a similar boundary response in internal waves reflecting from critical slopes; most of the internal wave energy is dissipated near to the boundary. We use experiments with a Reynolds number, $Re \sim 1000$, and numerical simulations that solve the full Navier-Stokes equations in the Boussinesq limit with $Re \sim 10^4 \sim 10^5$. Our data show that the rate of energy dissipation at the critical angle remains high even when viscous effects are minimized in the simulations. We also present laboratory data showing reflection from a turning depth and find high rates of energy dissipation though the no-slip boundary condition is not present. The data suggest that critical reflection dissipates energy from the internal wave field at high Reynolds numbers, and therefore, may contribute to the erosion of continental slopes.

1Louis Stokes Alliances for Minority Participation

9:16AM Q39.00008 Internal Wave Breaking in an Isothermal Atmosphere, DANIEL LECOANET, Princeton University, YUBO SU, DONG LAI, Cornell University — We present a series of two-dimensional numerical simulations of internal wave breaking in an isothermal atmosphere. Waves with a fixed frequency and wavenumber are continuously excited at the bottom of the atmosphere. As they propagate upward, their amplitude increases until they become nonlinear and break. The waves deposit their momentum and spin-up the upper layers of the atmosphere until the mean velocity is equal to the waves’ phase velocity. Waves continue to break at the shear layer at the bottom of the spun-up fluid, further depositing their momentum. We find the shear layer descends as $\exp(-t)$, as the momentum flux is constant, but the density of the atmosphere increases exponentially with depth. For simulations with sufficiently high Reynolds number, we find about 50% of the wave flux is reflected by the shear layer, 10% is transmitted into waves in the spun-up fluid, and 40% is absorbed.

9:29AM Q39.00009 Comparison of Internal Wave Kinetic Energy Estimates in Synthetic Schlieren and Particle Image Velocimetry1, KYLE HAKES, ALLISON LEE, ANNIE WESOLEK, JULIE CROCKETT, Brigham Young University — Synthetic Schlieren (SS) and Particle Image Velocimetry (PIV) are commonly employed experimental methods for investigating internal wave generation and propagation. PIV allows direct calculations of velocity and therefore kinetic energy, but at a relatively high setup cost. SS is a less expensive experimental method that is generally more easy to implement, but is not a direct measurement of velocity but instead natural frequency changes. This can be used to estimate kinetic energy but calculations are subject to the WKB approximation. Experiments were performed to investigate when the kinetic energy estimates from SS are a good approximation of the kinetic energy calculated from PIV for the same experiment. Internal waves generated by tidal flow over 4 different topographies in 4 different density profiles are explored via both SS and PIV methods for comparison. Preliminary findings indicate SS and PIV match well overall far from regions where WKB assumptions fail, and show that their ability to generate similar results depends on the shape of the topography and density profile.

1NSF Grant CBET-1606040

Tuesday, November 26, 2019 7:45AM - 9:16AM – Session Q40 Jets: Impinging 6b - Ahmed Naguib, Michigan State University

7:45AM Q40.00001 Instability of micro jet impinging onto a pool, MAOYING ZHOU, School of Mechanical Engineering, Hangzhou Dianzi University, BO LI, State Key Laboratory of Hydroscience and Engineering, Department of Energy and Power Engineering, Tsinghua University, JUN ZOU, The State Key Lab of Fluid Power & Mechatronic Systems, Zhejiang University — A micro liquid jet discharged into a downstream pool exhibits different instabilities at certain heights. For some nozzle heights, the water jet shows a steady wave profile while for some other heights, the water jet oscillates around the nozzle axis at given frequency. A series of experiments are conducted to identify the regimes of jet state with respect to nozzle heights, Reynolds numbers and Weber numbers. For the oscillation regime, oscillating characteristics of the jet are investigated in terms of different liquid properties. A simple model is developed to describe and explain the phenomena.

7:58AM Q40.00002 Impinging Tone Identification of Under-expanded Impinging Jets by Large Eddy Simulation1, MINGHANG LI, The University of Melbourne, SHAHRAH KARAMI, Laboratory for Turbulence Research in Aerospace & Combustion (LTRAC), Monash University, RICHARD SANDBERG, The University of Melbourne, JULIO SORIA, Laboratory for Turbulence Research in Aerospace & Combustion (LTRAC), Monash University, ANDREW OOI, The University of Melbourne — The acoustic and hydrodynamic feedback mechanism to predict discrete tones was first proposed by Powell (J.Acoust.Soc.Am., vol.83(2), 1988, pp.515533). The mechanism consists of the receptivity of the shear layer at the nozzle lip as well as the acoustic contribution from the downstream sources. The first part of the mechanism has been commonly accepted, while the locations of the downstream sources are still under debate. To further understand the mechanism, this work aims to identify the locations of the impinging tones by a novel methodology utilising the cross correlation and a ray tracing method. Each potential source is found with a certain ray tracing back to the nozzle exit. Joint probability density functions are then used to identify the impinging tones. Since the mean temperature out of the main jet plume and the wall jet varies little, another simplified method that does not consider refraction effects is proposed. Data from a recent Large Eddy Simulation is used to validate the current methods. The identified source locations are quantitatively determined and consistent with the second maximum of the root mean square of the pressure on the wall.

1Australian Research Council
8:11AM Q40.00003 Volumetric flow measurements of impinging jet on circular cylinder using STB1. MIRAE KIM, EUNSEOP YEOM, Pusan National University, MATTEO NOVARA, DANIEL SCHANZ, REINHARD GEISLER, JANOS AGOCS, ANDREAS SCHROEDER, German Aerospace Center (DLR), KYUNG CHUN KIM, Pusan National University

— Jet impingement is a direct and efficient way to transfer heat and mass in various applications. In practical applications, most jet flows are impinging on curved surfaces, however, less attention has been given to circular jet impingements on convexly curved surfaces. Interactions of three-dimensional flow structures of a round jet impinging obliquely on a convex circular cylinder was studied using high-resolution volumetric flow measurements by dense 4D Lagrangian particle tracking using the Shake-The-Box method and data assimilations by FlowFit. The Lagrangian stochasticized 3D flow field confirmed that the 3D curved wall jet spreads widely in spanwise direction after impingement then merged to the jet centerline downstream. The Coanda effect on 3D wall jet flow along the cylinder wall is vividly shown with the delay of separation up to 180 degrees. The strong shear layer near the impingement area produces large-scale vortices with high vorticity. These structures distribute throughout the surface and break down to multiple vortex structures with lower vorticity. Small-scale negative vortex structures are moved away from the wall jet and are sustained longer at the edge of the wall jet.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A2B2007117).

8:24AM Q40.00004 Effect of Bi-Modal Excitation of an Impinging Jet on Cooling of a Heated Impingement Surface1. BASIL ABDELMELGIED, AHMED NAGUIB, Michigan State University — Impinging jets have many engineering applications, such as heating, cooling, and drying. This work is part of a larger study focused on using different active flow control strategies for enhancement of the cooling effectiveness of impinging jet arrays. Here, we examine the influence of bi-modal acoustic forcing on the Nusselt number ($Nu$) distribution resulting from an axisymmetric jet impinging on a heated flat surface. The forcing scheme utilizes two concurrent sinusoidal waves, at the jet’s shear layer fundamental and sub-harmonic frequencies, to take advantage of the jet’s sub-harmonic resonance. The $Nu$ distribution is measured using temperature-sensitive paint applied to a heated stretched stainless steel foil. Data are obtained for jet Reynolds number based on jet diameter of 4000, jet-exit-to-plate distance range of 2 to 4 diameters, and different forcing parameters. The results illustrate the influence of the control on the jet’s cooling effectiveness and the dependence of this influence on the flow and the forcing parameters. Flow visualization is used to examine associated changes in the flow structure.

1This work is supported by NSF grant number CBET-1603720.

8:37AM Q40.00005 Transient model for characterizing the erosion mixing of stratified layer and turbulent impinging jet with both density and pressure gradients1. WOOYOUGN LEE, SIMON SONG, Institute of Nano Science and Technology, Hanyang University, Seoul, Republic of Korea, YOUNG SU NA, Dept. of Thermal Hydraulics and Severe Accident Research, Korea Atomic Energy Research Institute, Daejeon, Republic of Korea — The erosion mixing phenomenon of a stratified hydrogen layer caused by a turbulent impinging jet determines the distribution and mixing characteristics of hydrogen gas in a containment building of nuclear power plants (NPPs) during a severe accident. The mixed hydrogen gas can explode when in contact with ignition sources. To prevent the risk of the hydrogen explosion, it is necessary to quantitatively analyze the transient erosion mixing process over a long period of time. We experimentally and theoretically investigate the long-term erosion mixing process by the interaction between a stratified layer and a turbulent impinging jet. As a result, we propose a transient model for predicting the interface displacement of the stratified layer and the mean axial velocity and half width of the jet. We found that the prediction accuracy strongly depends on the consideration of the density and pressure gradient of the stratified layer and the jet. The results show that the predictions are in good agreement with the experimental data.

1This work was supported by the Korea Institute of Energy Technology Evaluation and Planning (KETEP) granted financial resource from the Ministry of Trade, Industry & Energy, Republic of Korea (No. 20184010201710).

8:50AM Q40.00006 Mixing of cold jets in cross flow into exhaust gases for cryogenic CO2 capture. ROBERT DIBBLE, FRANCISCO HERNANDEZ PEREZ, HONG IM, King Abdullah University of Science and Technology — For over a century, the widely accepted route for removal of CO2 from a gas stream has been absorption by amines. The liquid amine is sprayed downward in a vertical tower in which the exhaust stream is coming upward. In this counter-current flow, the falling amine droplets absorb CO2. The CO2-rich droplets collected at the bottom of the tower are pumped to a boiler, after which pure CO2 is extracted by heat. A new emerging process is the cryogenic carbon capture (CCC) process, in which the exhaust stream is cooled to near -90C and a cold fluid, such as methane, at -150C, is injected into the exhaust stream. The cold methane mixing with the exhaust gases forms CO2 in the “dry ice snow” form as the temperature rapidly descends below the sublimation point of CO2 (about -100C). The CO2 snow is easily collected. The methane as methane, at -150C, is injected into the exhaust stream. The cold methane mixing with the exhaust gases forms CO2 in the “dry ice snow” form as the temperature rapidly descends below the sublimation point of CO2 (about -100C). The CO2 snow is easily collected. The methane can be injected into the exhaust duct, from the wall, creating a classic “jet in cross flow” configuration. We find that more rapid mixing occurs if the methane is injected at 45 degree angle to duct wall, in both the flow direction and orthogonal to flow direction. The present study aims to explore the effect of the two angles on the mixing effectiveness. Simulations using LES show that most rapid mixing is achieved by the 45/45 degrees configuration.

9:03AM Q40.00007 Liquid Jet Impingement Cooling on Superheated Superhydrophobic Surfaces1. JACOB BUTTERFIELD, BRIAN IVERSON, DANIEL MAYNES, JULIE CROCKETT, Brigham Young University — Superhydrophobic (SH) surfaces form air cavities between nano- or micro-structures, resulting in self-cleaning properties. This is a potential solution for fouling in cooling applications, but the air cavities also impede heat transfer. Here, water jet impingement heat transfer on surfaces of varying microstructures and wettability is experimentally explored. Silicon wafers with micro-scale posts etched in a square array were used to create a range of SH surface microstructures. The SH surfaces are moved away from the wall jet and are sustained longer at the edge of the wall jet.

1This work was supported by the National Science Foundation [grant number: CBET-1707123].

Tuesday, November 26, 2019 7:45AM - 9:42AM – Session Q41 Advances in CFD Algorithms II 6c - Kursat Kara, Oklahoma State University
7:45AM Q41.00001 Using Evolutionary Neural Networks to Adapt the Jacobi Iterative Method for the Pressure Poisson Equation in a Multiphase Flow, TIANKUI XIAO, Pennsylvania State University, YIPENG SHI, Peking University, XIANG YANG, Pennsylvania State University — We utilize evolutionary neural networks (ENN) to determine a set of relaxation factors for the Jacobi iterative method, which is subsequently used to solve the pressure Poisson equation of a multiphase flow. The density of the fluid and the gas differ by a factor of 1000. An iterative method usually needs to be adapted to account for the large density difference in the field, which usually involves a long process of trial and error. In this talk, we show that ENNs could shorten this process. The iterative method we adapt is the Jacobi method. This method is embarrassingly parallel, and converges at a rate if employing a set of appropriately picked relaxation factors. Our ENN uses the iterative solution at one instance as the input, and determines the relaxation factor for the next iteration. Despite the local nature of our input to the neural network, we show that the results of the ENN is very close to the global optimum.

7:58AM Q41.00002 Automatic differentiation of a spectral difference code for sensitivity analysis1, JOSE I. CARDESA, CHRISTOPHE AIRIAU, Institut de Mécanique des Fluides de Toulouse — The computational fluid dynamics code JAGUAR, developed jointly by ONERA (Toulouse) and CERFACS, is a high-order code based on spectral differences and intended for unsteady aerodynamic simulations. In order to extend its use for shape optimization and flow control, it is convenient to adapt the code so that computing flow sensitivities is an efficient yet flexible process that can be adapted to very different problems. For this reason, a fully discrete approach was chosen that relies on automatic differentiation. A test case was analyzed to validate our approach in an unsteady problem, allowing us to identify key modifications to be implemented on the code so as to streamline the differentiation process and ease its replication in other problems. Tangent and adjoint modes were used to differentiate the parallel version of the code with TAPENADE and the adjoinable MPI library. Execution times and coding strategies will be provided to illustrate the benefits and drawbacks of the different approaches.

8:11AM Q41.00003 Stability and Accuracy of Semi-Extrapolated Finite Difference Schemes, ANDREW BRANDON, SHEILA WHITMAN, MIKAYLA FELDBAUER, NARSHINI GUNPATH, Lycoming College, BRENDAN DRACHERL, Rochester Institute of Technology, CARTER ALEXANDER, Lycoming College, LUCAS WILKINS, Vanderbilt University — When numerically solving partial differential equations, finite difference methods are a popular choice. Several factors contribute to the success of a finite difference method, including stability, accuracy, and computational cost. In response to the small stability regions of explicit methods and the computational cost of implicit methods, we have developed a novel discretization technique called semi-extrapolation. Semi-extrapolation generates explicit schemes from implicit schemes by applying extrapolation in an unconventional fashion. Unlike extrapolation, which can severely curtail stability, semi-extrapolation can improve stability, as compared to analogous explicit methods. Furthermore, semi-extrapolation can have unexpected effects on accuracy. In this presentation, we will introduce this semi-extrapolation and introduce two semi-extrapolated discretizations of the Advection-Diffusion Equation will be discussed. Then, the accuracies and stabilities of these semi-extrapolated discretizations will be compared to the accuracies and stabilities of analogous mainstream finite difference discretizations.

8:24AM Q41.00004 Solving the Navier-Stokes governing equations through quantum computing, FRANK GAITAN, Laboratory for Physical Sciences — We present a quantum algorithm that solves an arbitrary set of coupled non-linear partial differential equations and show how it can be used to solve the governing equations for a Navier-Stokes fluid. To test the algorithm we examine the problem of inviscid, compressible flow through a convergent-divergent nozzle. We numerically simulate application of the algorithm to find the steady-state flow when a shock-wave is and is not present in the divergent part of the nozzle. In each case excellent agreement is found between the output of the quantum simulation and the exact analytical solution, with the simulation successfully capturing the shock-waves in the latter case. We compare the computational cost of this quantum algorithm to that of deterministic and random classical algorithms; discuss future applications, as well as the potential long-term significance of quantum computing for the fluid dynamics community.

8:37AM Q41.00005 A finite volume scheme for stochastic PDEs in the context of fluctuating hydrodynamics1, SERGIO P. PEREZ, ANTONIO RUSSO, MIGUEL A. DURAN-OLIVENCIA, PETER YATSYSYIH, JOSÉ A. CARRILLO, SERAFIM KALLIADASIS, Imperial College London — The description of soft matter systems out of equilibrium requires the inclusion of fluctuations in the standard hydrodynamic equations for the evolution of conserved quantities. The associated general framework was postulated phenomenologically by Landau et. al., yielding what is known as Landau-Lifshitz fluctuating hydrodynamics. However, the numerical applicability of the fluctuating hydrodynamics entails several challenges which still remain elusive. In particular, conservative fluctuations, i.e. stochastic fluxes under the gradient operator, need to be properly accounted for. Besides, even for the simplest limit of these equations (which corresponds to the stochastic diffusion equation), the presence of a normally-distributed flux in the time-evolution equation for the density involves non-positive solutions, which are clearly unphysical. Hence the need for a robust method capable of handling stochastic fluctuations properly. Here we present a finite-volume scheme for stochastic gradient flows with nonlinear energy functionals, based on a hybrid upwind-central discretisation of both the deterministic and stochastic fluxes. The positivity of the density is ensured by an innovative time-adapting procedure based on the concept of Brownian trees. We exemplify the applicability and versatility of our method by solving the FHH in a wide spectrum of physical settings.

8:50AM Q41.00006 A symbolic regression approach for the development of high accuracy defect correction schemes1, HARSHA VADDIREDDY, OMER SAN, Oklahoma State University — Discrete equations are used to solve partial differential equations. Using higher order numerical schemes, we can traditionally reduce the number of grid points while preserving similar level of accuracy. Although high accurate numerical schemes can be constructed by using higher order polynomials, symmetry preservation, Pad approximation or Richardson extrapolation, it is well known that simple schemes sometimes completely eliminate the induced numerical errors when discretization parameters are chosen appropriately (e.g., $dt = dx/a$ for the first order Euler upwind scheme solving the linear wave equation). Furthermore, closure approaches are often introduced to account for nonlinear interactions in discrete models. To this end, we introduce a modular symbolic regression framework for finding optimal parameters, defect correction or closure terms, if necessary, to improve the accuracy of the underlying numerical procedures. Several examples are conducted to assess the feasibility of the proposed approach.

1Funded by the People Programme (Marie Curie Actions) of the European Unions Seventh Framework Programme (FP7/2007-2013) under REA grant agreement n. PCOFUND-GA-2013-609102, through the PRESTIGE programme coordinated by Campus France. Funded by the STAE Foundation through project 3C2T, managed by the IRT Saint-Exupery.

1This material is based upon work supported by the U.S. Department of Energy, Office of Science, Office of Advanced Scientific Computing Research under Award Number DE-SC0019290.
9:03AM Q41.00007 An Unconditionally Energy-Stable Scheme for Incompressible Flows with Outflow/Open Boundaries1, SUCHUAN DONG, XIAOYU LIU, Purdue University — We present an unconditionally energy-stable scheme for simulating incompressible flows on domains with outflow/open boundaries. The scheme combines the generalized Positive Auxiliary Variable (gPAV) approach and a rotational velocity correction type strategy, and the adoption of the auxiliary variable simplifies the numerical treatment for the open boundary conditions. The scheme admits a discrete energy stability property, irrespective of the time step sizes. Within each time step the scheme entails the computation of two velocity fields and two pressure fields, by solving an individual de-coupled Helmholtz (including Poisson) type equation with a constant pre-computable coefficient matrix for each of these field variables. The auxiliary variable, being a scalar number, is given by a well-defined explicit formula within a time step, which ensures the positivity of its computed values. We present numerical results with several flows involving outflow/open boundaries in regimes where the backflow instability becomes severe to demonstrate the performance of the method and its stability at large time step sizes.

1This work was partially supported by NSF

9:16AM Q41.00008 Spectral Methods for Time Series Prediction with Application to Fluid Flows, HENNING LANGE, STEVEN BRUNTON, NATHAN KUTZ, University of Washington — Forecasting the behavior of complex dynamical systems has a rich history in fluid dynamics. The Navier-Stokes equations and the dynamics of velocity fluctuations around the resulting base velocity profile are studied using the eddy-viscosity enhanced linearized equations. We utilize the second-order statistics of velocity fluctuations resulting from the stochastically forced periodic surface corrugation on skin-friction drag in turbulent channel flow. The effect of surface corrugation is modeled as a volume penalization of the time step sizes. Within each time step the scheme entails the computation of two velocity fields and two pressure fields, by solving an

9:29AM Q41.00009 On preserving accuracy of underlying discretization in overset meshes for incompressible flow1, ASHESH SHARMA, SHREYAS ANANTHAN, MICHAEL SPRAGUE, National Renewable Energy Laboratory, JAYANARAYANAN SITARAMAN, Parallel Geometric Algorithms, LLC — To accurately model the flow dynamics around wind turbines, it is crucial to capture well the many complex moving geometries involved. The need to resolve flow structures around these moving components motivates our choice of overset grids. Exchange of solution between the overlapping meshes is at the core of any overset framework. For incompressible-flow solvers, the associated linear systems arising at each time step can be solved in a coupled or a decoupled and iterative manner. In the former, monolithic linear systems are assembled and the overlapping meshes are coupled through constraint equations. In a decoupled solve, linear systems are created for each mesh, and information is exchanged at overset interfaces as a separate step after the governing equations have been solved on the individual meshes. This work examines cost and accuracy comparisons between overet coupled and decoupled solves using elliptic and hyperbolic systems representative of the incompressible Navier-Stokes equations.

1This work was supported by the U.S. Department of Energy under Contract No. DE-AC36-08GO28308 with the National Renewable Energy Laboratory

Tuesday, November 26, 2019 7:45AM - 9:29AM
Session Q42 Boundary Layers: Superhydrophobic Surfaces and Drag Reduction 6e - Julien Landel, University of Manchester

7:45AM Q42.00001 Model-based analysis of turbulent drag reduction in channel flow over corrugated surfaces, WEI RAN, University of Southern California, ARMIN ZARE, The University of Texas at Dallas, MHAHAO JOVANOVIC, University of Southern California — We develop a model-based approach to quantify the effect of spatially periodic surface corrugation on skin-friction drag in turbulent channel flow. The effect of surface corrugation is modeled as a volume penalization on the Navier-Stokes equations and the dynamics of velocity fluctuations around the resulting base velocity profile are studied using the eddy-viscosity enhanced linearized equations. We utilize the second-order statistics of velocity fluctuations resulting from the stochastically forced periodic surface corrugation on skin-friction drag in turbulent channel flow. The effect of surface corrugation is modeled as a volume penalization of the time step sizes. Within each time step the scheme entails the computation of two velocity fields and two pressure fields, by solving an

7:58AM Q42.00002 Retention of infused liquid for sustenance of drag reduction of turbulent flow over liquid infused surface, MARTAND MAYUKH GAMIELLA, EDGARDO JAVIER GARCIA CARTAGENA, STEFANO LEONARDI, The University of Texas at Dallas — Liquid infused surfaces (LIS) are composed of functionalized surface textures wetted with an immiscible, chemically-matched liquid lubricant. It has been experimentally demonstrated that grooved LIS configurations can reduce turbulent drag up to 35%. However, in practical configurations, due to high shear exerted by the flow, the infused liquid may get washed away. Previous studies have considered longitudinal bars. However, this texture cannot hold the lubricant-gas in the cavities. An alternative would be to place transversal bars to retain this lubricant. Direct numerical simulations of the flow in a channel with a rectangular mesh on the lower wall have been carried out. The aspect ratios of the cavities, the Reynolds and Weber numbers were varied. Also, two viscosity ratios between the two fluids, N=0.02 and N=0.4 were used to mimic idealized superhydrophobic and liquidinfused surfaces. In comparison to the flow over longitudinal bars, the addition of transverse bars reduces drag reduction. However, it was observed that the flow recirculates within the cavities with an expected reduction in drainage. In addition, at a higher Weber number, the interface above the mesh texture is more stable than that over longitudinal bars.
8:11AM Q42.00003 Investigation of turbulent flows over superhydrophobic surfaces with oscillatory slip length\(^1\), KIMBERLY LIU, ALI MANI, Stanford University — Superhydrophobic surfaces have been widely studied for the purposes of drag reduction; the formation of an air film on the surface allows for a slip boundary condition, greatly decreasing the wall friction and thus the overall friction drag. Experimental results of patterned superhydrophobic surfaces, with dynamic pressure control allowing for oscillation of the individual air films, show an even further drag reduction. In addition, it has been observed that under such conditions, strong oscillatory velocities are induced in the streamwise direction, reminiscent of a Stokes boundary layer (Wang and Gharib, Bull. Am. Phys. Soc. 2018). We present a numerical study that isolates two possible factors that we believe may be contributing to the significant drag reduction. We utilize direct numerical simulations (DNS) to study the effects of (1) the slip boundary condition with slip length oscillating in time and (2) oscillating streamwise wall boundary condition with nonzero bulk flow, revealing the foundational interplay between effective slip length, oscillations of slip length, and reduction of drag.

\(^1\)Supported by ONR

8:24AM Q42.00004 Impact of surfactant on the drag-reduction potential of superhydrophobic surfaces in turbulent flows\(^1\), JULIEN R. LANDEL, University of Manchester, SCOTT SMITH, FERNANDO TEMPRANO-COLETO, University of California Santa Barbara, FRANCOIS PEAUDECERF, ETH Zurich, FREDERIC GIBOU, PAOLO LUZZATTO-FEGIZ, University of California Santa Barbara — Recent studies, Peaudecerf et al. (PNAS 2017) and Song et al. (PRF, 2018), have shown the negative effect of surfactant on the drag-reduction performance of superhydrophobic surfaces (SHS) in laminar flow conditions. As SHS could have a large impact in reducing energy utilisation for many internal and external flow applications, particularly under turbulent flow regimes (e.g. ships, pipelines), it is important to understand and predict how surfactant-Marangoni stresses affect turbulent flows over SHS. This is crucially important, since surfactants are present in normal environmental conditions for most applications. Our existing model for SHS inclusive of surfactant (Landel et al. arXiv:1904.01194, 2019) captures the effect of surfactant for two-dimensional laminar flow. Using a technique inspired by Belyaev & Vinogradova (J. Fluid Mech. 652, 2010), we adapt our two-dimensional model to flows above three-dimensional SHS with longitudinal gratings. Then, we use the results of Fukagata et al. (Phys. Fluids 18, 2006) to relate the effect of the surfactant-affected slip length on the drag reduction of SHS in turbulent flows. We discuss the impact of the main parameters: the gas fraction, the surfactant concentration, and the Reynolds number on the drag reduction. Finally, we compare the results of our model with experimental results from the literature.

\(^1\)Supported by M. R. Prince Foundation and NSF XSEDE Allocation TG-CTS070067N.

8:37AM Q42.00005 Effect of Interface Deformation and Contact Line Motion on Drag Reduction with Superhydrophobic and Liquid-Infused Surfaces in Laminar and Turbulent Flow\(^1\), RAYHANEH AKHAVAN, AMIRREZA RASTEGARI, The University of Michigan — Effect of interface deformation and contact line motion on drag reduction (DR) with super-hydrophobic (SH) and liquid-infused (LI) surfaces is investigated in laminar and turbulent flow by direct numerical simulation (DNS) using a two-phase, single relaxation time, free-energy lattice Boltzmann method. In this method, the dynamics of a diffuse interface is incorporated into the governing equations using a Peng-Robinson free-energy functional. This obviates the need for interface tracking. DNS studies were performed in channel flows with longitudinal microgrooves of width \(\frac{g}{H} \leq 64\) or \(\frac{g}{H} \leq 32\). The results show that, in both laminar and turbulent flow, interface deformation and contact line motion can significantly modify the magnitude of DR compared to results obtained with ‘idealized’, flat SH or LI interfaces. The conditions under which the contact line depins and the interface breaks down are identified by DNS.

\(^1\)Supported by ONR.

8:50AM Q42.00006 A direct numerical simulation study of heat transfer over superhydrophobic and liquid-infused surfaces , UMBERTO CIRI, STEFANO LEONARDI, The University of Texas at Dallas — Recently, superhydrophobic (SHS) and liquid-infused surfaces (LIS) have been proposed as a wall treatment to achieve drag reduction in turbulent flows. Conceptually, these surfaces consist of a textured substrate with a secondary fluid filling the texture cavities over which the primary fluid flows. In the case of SHS, water flows over air trapped in the cavities, while for LIS a liquid lubricant is used instead of air. Turbulent drag reduction is possible because the second fluid creates a slippery interface with the primary fluid, thus reducing friction drag. While several studies have shown potential in terms of drag reduction, less attention has been dedicated to the heat transfer. The objective of this work is to study heat transfer characteristics of these surfaces and the correlation between the velocity and thermal fields (Reynolds analogy). Direct numerical simulations of turbulent flow and heat transfer are performed using different textured geometries (modeled with the immersed boundary method) and varying the viscosity ratio and interfacial tension between the two fluids. The level-set method is used to couple the dynamics of the interface between the two fluids to the Navier-Stokes equations.

9:03AM Q42.00007 Drag reduction variation with respect to changes in cavity geometry for a butterfly-inspired surface , SASHANK GAUTAM, AMY LANG, The University of Alabama — Monarch butterfly wings are covered with small scales arrange in a pattern that resembles roof shingles, with the tips pointing up forming low-profile cavities. As the flow passes over the wing, the skin friction drag depends on the direction of flow with respect to scale orientation. When the flow is transverse to the cavity orientation, a single vortex forms inside each of the cavity. Previous studies have documented this phenomenon as roller bearing effect which effects in sub-laminar drag for very low Re. Previous work focused on rectangular cavities with an aspect ratio of 2:1. This study aims to replicate the butterfly-inspired geometry with slanted wall cavities, specifically cavity models of AR 2:1 and AR 3:1 with cavity wall inclination angles of 22, 45 and 90 degrees, with the hope of optimizing the drag reduction. We hypothesize that models with AR 2:1 with 45 degrees cavity inclination angle and AR 3:1 with 22 degrees inclination angle will result in higher drag reduction. As all solid surfaces exert a no-slip condition, because of the formation of secondary vortices in the corners the primary embedded vortex will be in less contact with the cavity walls and thus be able to maintain a higher partial slip velocity as it interacts with the boundary layer.
Tuesday, November 26, 2019 9:46AM - 10:06AM –
Session R02 Francois Frenkiel Award for Fluid Mechanics talk: Static and dynamic fluid-driven fracturing of adhered elastica 6b - Neil Balmforth, University of British Columbia

Tuesday, November 26, 2019 9:46AM - 10:06AM –
Session R04 Andreas Acrivos Dissertation Award in Fluid Dynamics talk: The Development and Application of a Computational Method for Modeling Cellular-Scale Blood Flow in Complex Geometry 6e - Arezoo Ardekani, Purdue University
10:44AM S01.00002 Triad interactions induced by vortex shedding in a free jet in air. MARGERITA DOTTI, Department of Chemical and Biochemical Engineering, Technical University of Denmark, Lyngby, Denmark, PREBEN BUCHHAVE, Intaras Optics, Snderskovvej 3, 3460 Birkerød, Denmark, CLARA M. VELTE, Department of Mechanical Engineering, Technical University of Denmark, Lyngby, Denmark — The non-linear processes caused by the convection term in the Navier-Stokes equation are of fundamental importance for both the understanding of turbulence and modelling turbulent flows. These so-called triad interactions in Fourier space were investigated using a single Fourier mode injected into the initial part of a round, free jet in air. We studied the downstream velocity down the measured distance as a function of the target spectrum. Furthermore, the development of the downstream velocity was also calculated by means of a simple, one-dimensional computer model. The comparison between the measured power spectra and the computational ones showed a good agreement between them, allowing us to draw some interesting conclusions regarding the fundamental non-linear processes in turbulence.

10:57AM S01.00003 Transition to vortex shedding and its impact on heat and solute transport in membrane filtration systems. JINCHENG LOU, JACOB JOHNSTON, NILS TILTON, Colorado School of Mines, DR. TILTON’S RESEARCH LAB TEAM — We perform direct numerical simulations (DNS) to investigate the hydrodynamic stability of flow over a cylinder confined in a planar channel with a permeable wall. More specifically, we investigate how cylinder configuration influences transition to laminar vortex shedding, and how this impacts thermal and concentration boundary layers forming on the permeable wall. This flow plays an important role in modern desalination processes that use permeable membranes to remove solutes from feed solutions. The efficiency of these processes is reduced by the formation of thermal and solute boundary layers on the membrane surfaces. This has motivated considerable interest in disrupting these layers using nearby obstacles that generate vortex shedding. We show that while vortex shedding can indeed increase system efficiency by increasing transmembrane flow, it also generates recirculation zones on the membrane surface that lead to solute accumulation and precipitation. Such precipitation is known to damage membranes when treating complex feed solutions. The coupled momentum, energy, and mass transport equations are solved using a finite-volume method with recent advances in immersed boundary methods to enforce no-slip and no-flux conditions to second order spatial and temporal accuracy.

11:0AM S01.00004 Controlled symmetry breaking and vortex dynamics in intersecting flows. AMY SHEN, Okinawa Institute of Science and Technology, NOA BURSTEIN, SIMON HAWARD, Okinawa Institute of Science and Technology Graduate University — Vortices are a ubiquitous feature in complex flows and turbulence, but their dynamics are challenging to study due to their typically transient nature. Here, we perform a detailed study of the vortex dynamics and interactions associated with a symmetry-breaking flow instability at a 4-way intersection. By precisely controlling the flow rate above a critical value, we are able to induce the merging of two co-rotating vortices into a single structure and similarly to induce a single vortex to split into two. Using quantitative flow velocimetry, both processes are recorded with high spatial and temporal resolution. We find that both the merging and the splitting of vortices are exponential processes, with a rate that depends on the imposed flow rate. The vortex dynamics in our system are intimately connected with the symmetry-breaking transition and are affected by the degree of vortex confinement, which we control by varying the aspect ratio of the microfluidic device. We show how the confinement affects the fundamental nature of the flow transition, which varies from super through subcritical as the aspect ratio is increased. Our results are relevant to understand and predict flow transitions and vortex dynamics in flow interactions, particularly in confined environments.

11:23AM S01.00005 Vorticity dynamics for a spatially developing liquid jet within a co-flowing gas. WILLIAM A. SIRIGNANO, ARASH ZANDIAN, University of California, Irvine, FAZLE HUSSAIN, Texas Tech University, UCI COLLABORATION, TTU COLLABORATION — A three-dimensional transient round liquid jet with coaxial outer gas flow is simulated and analyzed via vortex dynamics. Two surface-deformation types separate at an indentation of the jet stem. Local vorticity explains the deformations in the recirculation zone behind the cap that affect the cap dynamics. The Kelvin-Helmholtz (KH) instability dominates the deformation region upstream of the cap (UR), unaffected by the behind-the-cap region (BCR). Different three-dimensional UR atomization mechanisms are delineated on a gas Weber number ($W_e^g$) versus liquid Reynolds number ($Re_l$) map, consistent with temporal studies and limited experiments, in a frame moving with the liquid velocity to portray better the similarity, avoiding the common misuse of velocity ratio. Vorticity distributions show periodic vortex development and surface deformation in the UR, with lost periodicity closer to the BCR. For practical density ratios and early times in the process, axial vorticity is mainly generated by baroclinicity while streamwise vortex stretching becomes more important later only at lower relative velocities with reduced pressure gradients. Pressure and viscous forces cause azimuthal acceleration. Azimuthal viscous forces are important even at high $Re_l$.

11:36AM S01.00006 Inflow vortex past moderately yawed cylinders. MOHAMMED KHAROUBA, JEAN-LOU PIERSON, IFP Energies nouvelles, JACQUES MAGNAUDET, IMFT, Universités de Toulouse — The flow past a finite-end yawed cylindrical particle is studied numerically. Three dimensionless parameters govern the problem when the flow is steady and uniform: the aspect ratio $\frac{L}{D}$ where $L$ is the length of the cylinder and $D$ its diameter, the yaw angle $\theta$ which is the angle between the cylinder axis and the inlet velocity, and the Reynolds number based on $D$. Particular attention is paid to the effect of these parameters on the particle wake and hydrodynamic loads. The aspect ratio is prescribed in the range $[0; 20]$, the yaw angle in the range $[0; 30]$, and the Reynolds number in the range $[0; 100]$. Various types of vortex patterns are observed, including steady shedding of two counter-rotating vortices, periodic shedding of counter-rotating vortices and unsteady shedding of hairpin-shaped vortices. Results show that the dynamical regime and time evolution of the loads change drastically with the yaw angle $\theta$. The wake is found to be unsteady in the range $Re \in [360; 400]$ at small yaw angles ($\theta \leq 30$) with $\frac{L}{D} = 2$. We propose a drag law valid for low and high Reynolds numbers in the case of a cylinder aligned with the flow.

11:49AM S01.00007 Hysteresis and Bistable Behavior in Low Reynolds Number Flow Over a Cylinder With a Slanted Afterbody. FERNANDO ZIGUNOV, PRABU SELLAPPAN, FARRUKH ALVI, FAMU-FSU College of Engineering — A cylinder with a slanted afterbody is a bluff body that has a wake pattern similar to aircraft fuselage wakes. Recent work (Bulathsinghala et. al. 2017; Zigorov et. al. 2019) improved our understanding of the vortex-dominated regime of this wake, where a pair of counter-rotating vortices is formed. Steepening the slant angle above a critical value causes the vortex-dominated wake to transition to a fully-separated ("stalled") wake, with a subsequent increase in unsteadiness and a significant drop in form drag. The current investigation shows the full three-dimensional flow topology of the two wake states, reconstructed through stacked stereoscopic particle image velocimetry (SPIV) and phase-averaged stereoscopic particle image velocimetry (S-SPIV). The wake transition to the bistable state is shown to be a function of a fixed slant angle and the wake presents a hysteretical behavior when the Reynolds number is slowly varied. The hysteresis effect is further detailed in this investigation through fully time-resolved, high-speed PIV, and physical insight into the flow instability mechanisms that contribute to this effect is presented.

1This research is made possible through the support of the Air Force Office for Scientific Research (AFOSR), grant no. FA9550-17-1-0228.
This research was supported in part by the Ford Foundation Post-Doctoral Fellowship, N00014-17-1-2068 under Dr. Timothy Bentley, and used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number ACI-1548562, through allocation TG-CTS120005.

This work is supported by the Office of Naval Research (ONR).

This work is supported by the Office of Naval Research.

This work is supported by the Office of Naval Research (ONR).
11:49 AM S02.00007 Neural-network-augmented Gaussian moment method for the statistics of cavitating bubble populations1. SPENCER BRYNGELSON, California Institute of Technology, ALEXIS CHARALAMPOPOULOS, THEMISTOKLIS SAPSIS, Massachusetts Institute of Technology, TIM COLONIUS, California Institute of Technology — Phase-averaged bubbly flow models are forced by moments of the bubble population dynamics. Computation of such moments requires a representation of the bubble population statistics. This has traditionally been accomplished via a classes method that evolves bins of discrete bubble sizes and computes the required moments via quadrature. We instead propose a method based upon explicit evolution of low-order moments of the bubble population and Gaussian closures. This circumvents the additional expense associated with the evolution of classes in lieu of a numerical evaluation of the associated closure integrals, which is particularly advantageous when the bubble population distributions are broad. This approach is exact for linear bubble dynamics, though has larger errors for bubble populations undergoing increasingly strong nonlinear dynamics. This problem is associated with the generation of higher-order moments, which we treat via recurrent neural networks. They are trained with Monte Carlo surrogate-truth data and augment our evolution of the low-order moments and evaluation of higher-order moments. The neural networks markedly improve model predictions, even for out-of-sample testing data.

1 United States Office of Naval Research Grant No. N0014-17-1-2676

12:02 PM S02.00008 Cavitation Collapse Near Slot Geometries, ELIJAH ANDREWS, IVO PETERS, University of Southampton — Vapor bubbles in water collapse towards a nearby solid boundary producing a jet that can clean, or damage, the boundary. It is useful to understand how different boundary geometries will affect the direction in which the jet is produced. The majority of research so far has focused on simple flat boundaries or limited cases with analytic solutions such as axisymmetric boundaries. We numerically and experimentally investigate how a slot in a flat boundary affects the jet direction of a single bubble. We use a boundary element model to predict how the jet direction depends on key parameters and show that the results collapse to a single curve when the parameters are normalized appropriately. We then experimentally verify the predicted dependencies using laser-induced cavitation and compare the experimental results to the predicted dependencies. This research provides useful insights into how jet direction is affected by slot geometries and demonstrates a method that can be used to investigate other complex geometries.

Tuesday, November 26, 2019 10:31 AM - 12:15 PM Session S03 Surface Tension IV 201 - Irmgard Bischofberger, MIT

10:31 AM S03.00001 Reverse Marangoni Propulsion of Disks and Hemispheres at Finite Reynolds Numbers1. SAMRAT SUR, University of Massachusetts - Amherst, HASSAN MASOUD, Michigan Tech, JONATHAN ROTHSTEIN, University of Massachusetts - Amherst — In this presentation, the experimentally observed phenomenon of Reverse Marangoni propulsion will be presented for both a thin cylindrical disk and a hemisphere floating on an air-water interface. Each particle was propelled by an asymmetric release of a surfactant to locally reduce the surface tension. Marangoni surfers typically propel themselves forward in the direction of high surfactant concentration. However, by systematically varying the water depth we will show that increasing confinement initially causes the velocity of the Marangoni surfer to slow, then come to rest and finally to reverse direction resulting in the Marangoni surfers moving in the direction of low surfactant concentration. Particle tracking and PIV measurements will be used to measure flow field induced by Marangoni flow beneath the disk and hemisphere and understand the origin of the reverse Marangoni flow. This phenomenon of reverse Marangoni flow has been predicted theoretically for Stokes flow at zero Reynolds number. We will show that the reverse Marangoni motion is not only dependent on the water depth confinement but also on Reynolds number. With increasing Reynolds number, increased confinement is needed to observed reverse Marangoni flow. These experimental results are in excellent agreement with the prediction of numerical simulations.

1 NSF CBET 1705519

10:44 AM S03.00002 Analytical study for vapor-driven solutal Marangoni flows inside a sessile droplet1. JUNIL RYU, JUNKYU KIM, JONGHYEOK PARK, HYOUNGSOO KIM, KAIST — Flow control inside a sessile droplet is important in microfluidic mixing, materials patterning and coating applications. Recently, a solutal Marangoni flow driven by a vapor of volatile liquid has been introduced as a novel way of flow controller and mixer, which does not require external devices and pollute mixing samples. In this talk, we will present the controlled flow patterns and efficient mixing inside a sessile droplet using solutal Marangoni effects. Furthermore, we developed a theoretical model to predict the vapor-driven solutal Marangoni flows. By matching the experimental and theoretical results, we estimate the profile of vapor distribution of volatile liquid, which is very difficult to directly measure from experiments. Using this analytical model, we further investigate how the boundary condition changes the internal flow pattern. Several possible cases will be discussed during the talk.

1This work was supported by Basic Science Research Program through the National Research Foundation of Korea funded by the Ministry of Science (NRF-2018R1C1B6004190) and BK21 Plus program. We also thank that this paper is based on research which has been conducted as part of the KAIST-funded Global Singularity Research Program for 2019.

10:57 AM S03.00003 Hydrodynamic Doppler effect in a weakly compressible two-dimensional medium. ILDOO KIM, None — This study is based on two experimental observations in flowing soap film channels. First, the vorticity field is strongly correlated with the thickness field. Second, the thickness field propagates at the Marangoni wave speed. Using the two observations, we propose and review the hypothesis that the vorticity field propagates at the Marangoni wave speed $c$. It is inferred from the hypothesis that a retarded hydrodynamic potential function can be solved in an approach similar to the Lienard-Wiechert potential of the relativistic electromagnetic theory, and the retarded potential implies an elongation effect of a vortex array by $1/(1 + v^2c^2)$ when the array recedes from the origin at $v$. The theory is compared with the experiment, and they agree within the margin of measurement error.

11:10 AM S03.00004 Marangoni-driven film climbing on a draining pre-wetted film, NAX XUE, Princeton University, MIN PACK, Baylor University, HOWARD STONE, Princeton University — In this experimental study, we report a Marangoni flow generated when a bath of surfactant contacts a pre-wetted film, which is set by gravitational drainage on a vertical substrate. High-speed interferometry is used to measure the front position of the climbing film and the film thickness profile, and the effect of the surfactant concentration and the pre-wetted film thickness on the film climbing is studied. As a result, higher surfactant concentration induces a faster and thicker climbing film. Also, for high surfactant concentrations, where Marangoni driving dominates, increasing the film thickness increases the rise speed of the climbing front since viscous resistance is less important. In contrast, for low surfactant concentrations, where Marangoni driving balances with gravitational drainage, increasing the film thickness decreases the rise speed of the climbing front while enhancing gravitational drainage. We rationalize these observations by establishing a model that analyzes the climbing front, either in the Marangoni driving dominated region or in the Marangoni balanced, drainage region. Our work highlights the possible effects of the gravitational drainage on the Marangoni flow, both by setting a pre-wetted film and by resisting the film climbing.
11:23AM S03.00005 Marangoni Instability and Interfacial Turbulence of a Water Drop at an Oil/Alcohol Interface¹, SAMI YAMANIDOZISORKHABI, GARETH H. MCKINLEY, IRMGARD BISCHOFF-BERGER, MIT — Water and ethanol are fully miscible, while anise oil and ethanol are partially miscible and anise oil and water are immiscible. Mixing of all three liquids, however, results in an equilibrium state emulsion of anise microdroplets in water/ethanol solution. Here, we study the mixing of a water drop at the interface of anise oil and ethanol until the equilibrium emulsion state is achieved. We use a combination of regular imaging and Schlieren imaging to visualize the mixing phenomenon. This mixing involves two processes: (i) introduction of a water drop at the anise oil/ethanol interface and formation of a vortex ring due to natural convection caused by exothermic mixing of water and ethanol, (ii) growth of the vortex ring at the interface due to Marangoni forces until the equilibrium state is achieved. We show that the inhomogeneities at the anise oil/ethanol interface apply non-uniform Marangoni forces on the vortex ring resulting in its deformation and a non-uniform distribution of the final mixing product, i.e., the equilibrium state emulsion, at the interface.

¹We acknowledge the support of the Natural Sciences and Engineering Research Council of Canada (NSERC). Cette recherche a été financée par le Conseil de recherches en sciences naturelles et en génie du Canada (CRSNG).

11:36AM S03.00006 Solutal Marangoni flow induced by a solute source. ISLAM BE-NOUAGUEF, Department of Mathematical Sciences, New Jersey Institute of Technology, Newark, New Jersey, NAGA MUNSUNURI, Department of Mechanical and Industrial Engineering, New Jersey Institute of Technology, Newark, New Jersey, DENIS BLACKMORE, Department of Mathematical Sciences, New Jersey Institute of Technology, Newark, New Jersey, IAN S. FISCHER, PUSHPELLDRA SINGH, Department of Mechanical and Industrial Engineering, New Jersey Institute of Technology, Newark, New Jersey — The study of the solutocapillary flow induced in a waterbody due to the presence of a solute source or sink on its surface is reported. The surface tension of water increases with increasing salt concentration, and so, for example, when a freshwater source is present on the surface the local salt concentration is reduced which in turn makes the interfacial tension near the source smaller than that away from the source where the salt concentration is larger. This gives rise to an interfacial gradient away from the source which drives the flow. We have analytically studied the axially symmetric analytic solution to this problem and have made a comparison with the experimental data obtained by the PIV (Particle Image Velocimetry) and PLIF (planar laser-induced fluorescence) techniques. It is shown that a freshwater source gives rise to a doublet flow such that the flow comes towards the source within a conical region with its vertex at the source and outside the conical region the flow moves away from the source. The half cone angle increases with increasing source strength and for a typical solutocapillary flow it is ~70-80 degrees.

11:49AM S03.00007 Influence of nonlinear temperature dependence of surface tension on longwave oscillatory Marangoni patterns. ALEXANDER MIKISHEV, Sam Houston State University, ALEXANDER NEPOMNYYASHCHY, Technion — In most theoretical papers on Marangoni convection the authors assume the linear dependence of the surface tension on temperature. However, according to experiments, that dependence is more complex. In the present work, we consider the influence of nonlinear temperature dependence of surface tension on the nonlinear dynamics of waves created by an oscillatory source within a conical region with its vertex at the source and outside the conical region the flow moves away from the source. We have analytically studied the axially symmetric analytic solution to this problem and have made a comparison with the experimental data obtained by the PIV (Particle Image Velocimetry) and PLIF (planar laser-induced fluorescence) techniques. It is shown that a freshwater source gives rise to a doublet flow such that the flow comes towards the source within a conical region with its vertex at the source and outside the conical region the flow moves away from the source. The half cone angle increases with increasing source strength and for a typical solutocapillary flow it is ~70-80 degrees.

12:02PM S03.00008 Surface tension-driven flows induced by polymerization waves¹, REDA TIANI, Universit Libre de Bruxelles, JOHN A. POJMAN, Louisiana State University, LAURENCE RONGY, Universit Libre de Bruxelles, DEPARTMENT OF CHEMISTRY (LOUISIANA STATE UNIVERSITY) COLLABORATION — Thermal frontal polymerization (FP) is a process in which a monomer-initiator mixture is converted into a polymer via a localized reaction zone that propagates due to the interplay between heat diffusion and exothermic polymerization whose reaction rate increases with temperature following Arrhenius’ dependence. Recent experiments considering horizontally propagating FP have evidenced the presence of hydrodynamic flows that interfere with the dynamics of polymerization waves and even possibly prevent their initiation. Since those experiments are conducted in systems open to the air, surface tension-driven (or Marangoni) flows, due to temperature gradients between the cold monomer-initiator mixture and the hot polymer solution, are expected to play an important role in the observed experimental results. In this work, we present a two-dimensional model that includes the oscillatory instability recently discovered in [1] in the limit of small Biot number Bi and wavenumber k, k ≈ Bi^{1/2}. Near the critical Marangoni number, that dependence is described by a Taylor series around the reference temperature value. The set of amplitude equations governing the nonlinear interaction of waves has been derived. The stability of different wave patterns and wave pattern selection are investigated. REFERENCE: [1] S. Shklyaev, A. Ababuzhev, and M. Khenner, Phys. Rev. E, 85, 016328 (2012).

¹The authors thank the Actions de Recherches Concertees program and the F.R.S.-FNRS for their financial support

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S04 Nano Flows 203 - Yuan-Nan Young, New Jersey Institute of Technology

10:31AM S04.00001 Correlations at Liquid/Solid Interfaces Relating Molecular Configurational Effects to the Kapitza Resistance¹, HIROKI KAI-FU, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd., MC 128-95, Pasadena, CA 91125 — Todays electronic systems for banking, medicine and transportation rely critically on ever more powerful integrated chips which can generate local power densities in excess of 100 W/cm² leading to catastrophic thermal failure. Such excess heat has become the limiting factor in information processing. Liquid cooling is therefore being used to mitigate this problem. Fundamental understanding of thermal resistance at liquid/solid (L/S) interfaces is therefore indispensable to future design. Interfacial thermal transport is normally quantified by the Kapitza resistance although its relation to phonon transport at L/S interfaces is still lacking. Computational studies have mostly focused on the effects of liquid wettability and contact density, whose enhancement favors formation of absorbed layers that lower the resistance. Using non-equilibrium molecular dynamic simulations of a monoatomic Lennard-Jones liquid confined between solid walls, we examine interface configuration effects by varying the intermolecular distance representing the L/S potential minimum. These studies reveal how configuration of the liquid and solid molecules at the interface in the absence of liquid flow controls the magnitude of the Kapitza resistance.

¹HK gratefully acknowledges support from a 2019 NASA Space Technology Research Fellowship.
10:44AM S04.00002 Nanoscale Capillary Bridges and the Role of Hydration Forces1
CARLOS COLOSQUI, SIJIA HUANG, Stony Brook University; YUAN YOUNG, New Jersey Institute of Technology; HOWARD STONE, Princeton University — This talk presents results from theoretical and numerical analysis of a nano water bridge (of height 1 to 10 nm) studied via both fully atomistic molecular dynamics (MD) simulations and continuum-based models based on the Young-Laplace equation. For nanoscale separations between two flat walls, surface forces (e.g., van der Waals and hydration forces) significantly affect the capillary bridge shape, as well as the liquid-solid contact area and contact angle. Nevertheless, the local radius of the capillary bridge is reasonably well described by the classical Young-Laplace equation for surprisingly small heights of about 3 nm (i.e., 10 molecular layers). On the other hand, the curvature predicted by the classical Young-Laplace equation is constant and differs significantly from that reported by MD simulations. As a result, when the water bridge height is smaller than 5-10 nm we observe large differences between adhesion forces obtained from MD simulations and those predicted by Young-Laplace. To accurately account for results from fully atomistic MD simulations we must extend the Young-Laplace description by including a disjoining pressure that considers hydration forces associated with molecular layering and structural changes in the water near the walls.

1This work is supported by the National Science Foundation (DMS-1614892 (CC), DMS-1614907 (HAS), DMS-1614863 (YNY))

10:57AM S04.00003 Effect of Charge Inversion on Nanoconfined Flow of Multivalent Electrolyte Solutions1
ANDRES ROJANO, ANDRES CORDOBA, Universidad de Concepcion, JENS HONORÉ WALTHER, Technical University of Denmark, HARVEY A. ZAMBRANO, Universidad Tecnica Federico Santa Maria — Miniaturized devices integrated by nanoparticles have great potential for clinical and biotechnological analysis due to amplified sensitivity, faster response and increased portability. The transport properties of an electrolyte solution flowing through a nanopore in which the electrical double layer occupies a considerable part of the cross section can be altered by the interfacial charge inversion (CI). Hence, an exhaustive understanding of the fluid transport in presence of CI is essential to develop more efficient nanofluidic devices. Here, molecular dynamics simulations of multivalent electrolyte solutions in silica nanochannels are conducted to study the effect of CI on hydrodynamic properties. The solutions consist of water as solvent, chlorine as co-ion and different shares of counter-ions i.e. sodium, magnesium and aluminum. From atomistic trajectories, we find that the magnitude of the effective viscosity is correlated to the concentration and valence of the counter-ions in the solution. Additionally, we show that the CI value is directly related to the hydration shell size of the counter-ions. Moreover, the results suggest that higher CI produces a gel-like region adjacent to the channel wall that increases the interfacial viscosity and friction coefficient.

1We thank funding from CONICYT scholarship 21181167, computational support from DTU.

11:04AM S04.00005 The impact of ligand-receptor binding on the rolling behavior of nanoparticles
HUILIN YE, ZHIQIANG SHEN, YING LI, University of Connecticut — The adhesive rolling of nano-sized particle (NP) plays an essential role in the delivery of therapeutic or imaging agents to diseased microvasculatures. We investigate the adhesive behaviors of NPs on a substrate under the shear flow. Based on the energy balance analysis, we theoretically derive the steady rolling equation for different sized NPs. Contrary to the fundamental Stokes prediction, it is found that smaller NPs move faster than their larger counterparts under the ligand-receptor binding (LRB) effect. Further, the hydrodynamic strength (quantified by shear rate γ) is demonstrated to be associated with the steady rolling velocity (v) of NPs as R = v/γ (R is radius of NP). This scaling is attributed to the size dependence of the adhesive kinetics that is described by energy based stochastic model. We also find the enlargement of flow strength will trigger the transition from adhesive rolling to free rolling of NPs, due to the saturation of stretching of biological bonds forming between ligands and receptors. The size dependence of the rolling behavior may provide a guidance for engineering efficient NPs in biomedical applications.

11:18AM S04.00006 Size determines the adhesive rolling of nanoparticle: smaller rolls faster
ANDRES ROJANO, ANDRES CORDOBA, Universidad de Concepcion, JENS HONORÉ WALTHER, Technical University of Denmark, HARVEY A. ZAMBRANO, Universidad Tecnica Federico Santa Maria — Design of efficient nanofluidic platforms requires effective reduction of flow resistance within the channel network. With this purpose, 2D-materials can be deposited on polymeric substrates increasing the transport efficiency of water solutions through nanopores. In the present work, we show that significant drag reduction can be achieved in a polyamide nanoslit pore by using graphene and boron nitride as wall coatings. Here, Molecular Dynamics simulations are performed to study water flow through uncoated and coated polyamide nanoslit pores. From atomistic trajectories, we investigate interfacial properties and evaluate the effect that the polymeric matrix has on water structure inside the pores. Furthermore, we compute density and temperature profiles, molecular orientations, friction coefficient and velocity profiles. Using these observables, we analyze the correlation between local water structure, flow enhancement and slip length. Our results indicate that in coated pores the interactions between water molecules and the underlying polyamide substrate have a significant influence on the flow rates. The insights reported in this work may assist the design of strategies to achieve low friction water transport in nanostructured pores.

11:36AM S04.00006 Size determines the adhesive rolling of nanoparticle: smaller rolls faster
HUILIN YE, ZHIQIANG SHEN, YING LI, University of Connecticut — The adhesive rolling of nano-sized particle (NP) plays an essential role in the delivery of therapeutic or imaging agents to diseased microvasculatures. We investigate the adhesive behaviors of NPs on a substrate under the shear flow. Based on the energy balance analysis, we theoretically derive the steady rolling equation for different sized NPs. Contrary to the fundamental Stokes prediction, it is found that smaller NPs move faster than their larger counterparts under the ligand-receptor binding (LRB) effect. Further, the hydrodynamic strength (quantified by shear rate γ) is demonstrated to be associated with the steady rolling velocity (v) of NPs as R = v/γ (R is radius of NP). This scaling is attributed to the size dependence of the adhesive kinetics that is described by energy based stochastic model. We also find the enlargement of flow strength will trigger the transition from adhesive rolling to free rolling of NPs, due to the saturation of stretching of biological bonds forming between ligands and receptors. The size dependence of the rolling behavior may provide a guidance for engineering efficient NPs in biomedical applications.
Simulation of the Canonical Shock-Turbulence Interaction, ROZIE ZANGENEH, Prairie View AM

Therefore can avoid Jacobian evaluation. For an accurate, non-oscillatory solution which unlike the existing methods, does not involve Riemann solvers or characteristic decomposition, the use of high-bandwidth schemes with minimal dissipation and dispersion. The existing methods such as ENO, WENO, and RKDG method schemes that can capture flow discontinuity at the shock while capturing broadband spatial and temporal variations in a turbulent flow suggests such as shocks and contact surfaces, in high-speed compressible flows with interactions of shear driven turbulence, requires dissipation numerical shock with adequately low dissipation for the minimum influence of Large Eddy Simulation (LES) of turbulent flows. To this end, high-resolution shock fronts from background noise. This algorithm is also capable of separating entangled shock fronts through pattern recognization, which extract the pixel locations of the shocks. The edge detection algorithm takes advantage of shock waves' light intensity feature to distinguish correction to any optical distortions is applied to the photographs. Next, noise removal and edge detection algorithms are implemented to large amount of schlieren photographs. Here we present an automated algorithm to track individual shock fronts and triple points. First, experimental studies of multiple shock wave interaction to study transition from regular to irregular reflection rely on the processing of a experimental models are supported by extensive molecular-dynamics simulations, as well as evidence from some directly comparable experiments.

Currently at Carnegie Mellon University

12:02PM S04.00008 Measuring the hydrodynamic wall position and viscoelastic friction coefficient by molecular dynamics1, TAKESHI OMORI, NAOKI INOUE, Osaka University, LAURENT JOLY, SAMY MERABIA, Claude Bernard University Lyon 1, YASUTAKA YAMAGUCHI, Osaka University — Flows in nanofluidic systems are controlled by the hydrodynamic boundary condition (BC), involving the friction coefficient and the hydrodynamic wall position. Here we considered a liquid nano-slab confined between two walls, where we derived, from the Stokes equation and the Navier slip BC, analytical expressions for the liquid response to an oscillatory tangential motion of the walls in terms of the wall shear stress and mean fluid velocity. By fitting these expressions to molecular dynamics simulation results, we could extract both the viscoelastic friction coefficient and hydrodynamic wall position for walls with three different wettabilities, hence fully characterizing the frequency-dependent hydrodynamic boundary condition. The proposed methodology applies to a variety of liquid-solid interfaces of interest, e.g. for flows of complex fluids or fluids at a low temperature. It should also support methodological developments on the characterization of the hydrodynamic slip in general, the further development of the quartz crystal microbalance measurement technique.

1This work was financially supported by JSPS KAKENHI Grant Nos. 18K03929 and 18K03978. YY was also supported by JST CREST Grant No. JPMJCR1811, Japan. LJ was supported by the ANR, Project ANR-16-CE06-0004-01 NECtAR and the Institut Universitaire de France. LJ benefited from a JSPS international fellowship for research in Japan.

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S05 Compressible Flows: Shock Interactions 204 — Dale Pullin, Caltech

10:31AM S05.00001 Evolution of perturbed planar shock waves1, DALE PULLIN, NAJJIAN SHEN, California Institute of Technology, RAVI SAMTANEY, King Abdullah University of Science and Technology, VINCENT WHEATLEY, University of Queensland — We consider the evolution of a planar gas-dynamic shock wave subject to smooth initial perturbations in both Mach number and shock-shape profile. A complex-variable formulation for the general shock motion is developed based on an expansion of the Euler equations proposed by Best [Shock Waves, 1, 4, (1991)]. The zeroth-order truncation of Best’s system corresponds to the equations of Whitham’s geometrical shock dynamics (GSD) while higher-order corrections provide a hierarchical description that can be closed at any order, as detailed initial flow conditions for the flow immediately behind the shock are prescribed. Solutions to the first- and second-order closure of Best’s system for the evolution of planar perturbations are explored numerically to investigate the development of a finite-time singularity in the shock shape profile. Results are compared to those obtained using GSD [Mostert et al., J. Fluid Mech., 246, (2016)].

1KAUST Office of Sponsored Research

10:44AM S05.00002 Image processing and edge detection techniques to quantify regular to irregular shock wave transition obtained from experiments1, LINGZHI ZHENG, presenter, BENJAMIN KATKO, BARRY LAWLOR, CLAIRE MCGUIRE, JANE ZANTESON, KEVIN NGUYEN, VERONICA ELIASSON, coauthor — Experimental studies of multiple shock wave interaction to study transition from regular to irregular reflection rely on the processing of a large amount of schlieren photographs. Here we present an automated algorithm to track individual shock fronts and triple points. First, correction to any optical distortions is applied to the photographs. Next, noise removal and edge detection algorithms are implemented to extract the pixel locations of the shocks. The edge detection algorithm takes advantage of shock waves’ light intensity feature to distinguish shock fronts from background noise. This algorithm is also capable of separating entangled shock fronts through pattern recognition, which utilizes a discretization method to reduce complex shock geometries to localized linear patterns. Collectively, the algorithms can track shock wave characteristics to sub-pixel precision. Extractable characteristics include positions and propagation velocities of shock fronts, vertical and horizontal velocities of the Mach stem, and triple point trajectories during shock interactions. This algorithm can process large volumes of data with minimal manual operations, making image processing more efficient and productive.

1We acknowledge the support from AFRL FA8651-17-1-004 and NSF CBET-1803592.

10:57AM S05.00003 On the Use of Non-Staggered Central Schemes for Large Eddy Simulation of the Canonical Shock-Turbulence Interaction, ROZIE ZANGENEH, Prairie View AM

The objective of this study is to investigate the ability of the semi-discrete, non-staggered central scheme to capture the shock with adequately low dissipation for the minimum influence of Large Eddy Simulation (LES) of turbulent flows. To this end, high-resolution LES simulations are performed to study the interaction of a stationary shock with fully developed turbulent flows. The presence of discontinuities, such as shocks and contact surfaces, in high-speed compressible flows with interactions of shear driven turbulence, requires dissipation numerical schemes that can capture flow discontinuity at the shock while capturing broadband spatial and temporal variations in a turbulent flow suggests the use of high-bandwidth schemes with minimal dissipation and dispersion. The existing methods such as ENO, WENO, and RKDG method typically involve Riemann solvers, characteristic decomposition and Jacobian evaluation, making them complex and difficult to implement in a collocated polyhedral framework. Here, a central scheme which developed by Nessyahu and Tadmor is introduced as an alternative approach for an accurate, non-oscillatory solution which unlike the existing methods, does not involve Riemann solvers or characteristic decomposition, therefore can avoid Jacobian evaluation.
11:10AM S05.00004 Phase Analysis of Disturbances within Transitional Shock Boundary Layer Interactions.~1 JAMES THREADGILL, JESSE LITTLE, University of Arizona — Two-dimensional laminar/transitional Shock Boundary Layer Interactions (SBLIs) have been investigated to assess the influence of Reynolds number and interaction strength, and to probe unsteadiness mechanisms. SBLIs are induced by various ramps (15° < θ < 28°) mounted to a flat plate in Mach 4 flow (Re/L = 4.6×10^6 m^{-1}) at various locations (1.2×10^5 < Reₐ < 2.5×10^5). Oil-flow visualization and high-speed schlieren (50 kHz) have been employed to characterize the flow. The naturally laminar incoming boundary layer experiences significant separation within the SBLI (24 < L/δ₀ < 40). Strong SBLIs with high Reynolds numbers initiate transition within the elongated separated shear layer, promoting unsteadiness. Low-frequency separation shock motion is observed, similar to strong turbulent SBLIs (Stₐ ≈ 0.03). This motion is incoherent with all upstream features, supporting claims of a downstream mechanism driving SBLI unsteadiness. Phase analysis shows that the reattachment shock motion precedes the shear layer which precedes the separation shock motion. In addition, an upstream propagating convective density disturbance was identified within the separated flow (u ≈ -0.2u∞) which directly influences subsequent motion of the separation shock. These observations provide vital input when characterizing complex turbulent unsteady SBLI mechanisms.

1This investigation has been made possible by the support of the Raytheon Company and is greatly appreciated.

11:23AM S05.00005 A Compression - Ramp Shock/Boundary-Layer Interaction Over a Compliant Panel at Mach 5.~1 MUSTAFA MUSTA, LEON VANSTONE, MARC EITNER, JAYANT SIROHI, NOEL CLEMENS, University of Texas at Austin — A compression ramp induced shock wave/boundary layer interaction over a rigid and compliant surface was studied in a Mach 5 flow using high-speed stereo digital image correlation (DIC) and high-speed pressure-sensitive paint (PSP). The compliant panel, made of polycarbonate, is 2 mm thick and gives a fundamental first three mode frequency of about 700, 949, and 1350 Hz. The compression ramp, located at the downstream end of the compliant panel, is the compression angle of 28-degree compression ramp with 1-inch fence distance. The DIC will give time-resolved measurements of the displacement of the compliant panel, and the PSP will give the surface pressure over the entire panel. The simultaneous high-speed pressure and displacement measurements will allow analyzing the structural response of the compliant panel, flow unsteadiness, and the shock-foot in frequency and the time domain and compare with the rigid panel case.

1This study is sponsored by Sandia National Laboratories

11:36AM S05.00006 Thermally driven unsteadiness of Shock Boundary Layer Interaction SANG LEE, BRIAN ROMERO, University of New Mexico, TONGHUN LEE, University of Illinois at Urbana-Champaign — Deficiencies still remain in understanding the unsteadiness associated with the shock interacting with the incoming turbulent boundary layer flow exposed to thermal variation which is caused by the difference in the wall temperature and that of the ambient flow. The present study examines the effects of wall temperature on the dynamics of the unsteady separation bubble in a shock boundary layer interaction region at various Mach numbers. High order numerical simulations of a compression ramp are validated with existing experimental and numerical data at low Mach numbers, then various ratios of wall temperature to recovery temperature are investigated for increasing Mach numbers.

11:49AM S05.00007 Predictability of wall-modeled large-eddy simulation for shock wave/turbulent boundary layer interacting separated flows1 YUMA FUKUSHIMA, SOSHI KAWAI, Tohoku University — In this talk, we first discuss the important flow features (such as separation and reattachment) of the shock wave and turbulent boundary layers interaction based on wall-resolved large-eddy simulation (WRLES) database. By using the WRLES database, we then investigate the capability of the wall-stress-model-based equilibrium and non-equilibrium wall-modeling in LES [Kawai and Larsson, PoF 2012, 2013] for predicting the boundary layer separation and reattachment induced by a strong pressure gradient of the shock wave. In this study, we focus on non-equilibrium terms in the streamwise momentum equation (pressure gradient and convection terms) that are important in the flow separation and reattachment and investigate how these terms are treated in the wall-modeled LES (WMLES). We also discuss the WMLES of transonic airfoil buffet phenomena at high Reynolds number, where the unsteady shock wave induces the separation and reattachment.

1This work was supported in part by MEXT as a social and scientific priority issue to be tackled by using post-K computer. A part of this research used computational resources of the K computer provided by the RIKEN Advanced Institute for Computational Science (Project ID:hp150254,hp160205,hp170267,hp180185,hp180158,hp190164).

12:02PM S05.00008 Direct numerical simulations (DNS) to study the effect of particle motion and Reynolds number on the drag force during shock-particle interactions YASH MEHTA, JONATHAN D. REGELE, Los Alamos National Laboratory — The large disparity between length and time scales associated with explosive dispersal of particles makes the numerical study of these flows extremely challenging. With the recent availability of large-scale computational resources, the number of particle-resolved studies are increasing but most neglect particle motion, viscous effects or both. In the present study, we are interested in studying the effect of viscosity as well as motion on the drag forces experienced by a cluster of spherical particles during shock interaction. We are also interested in studying the anomalous drag reported by Bordoloi et al. (JFM Rapids 2017) in their experiments of a shock interacting with a dilute cloud of micro-particles. Towards this end, as a first step, we perform a direct numerical simulation of a spherical particle under shock-wave loading by solving the Navier-Stokes equations. We use an adaptive wavelet collocation method solver with volume penalization IBM for the particles.LA-UR-19-27293

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S06 Acoustics: Aeroacoustics II 205 - Aaron Towne, University of Michigan
10:31AM S06.00001 Aeroacoustic source localization and source level estimation during wind tunnel testing of a flat plate with and without a gap. ALEXANDER DOUGLASS, Mechanical Engineering, University of Michigan, NATASHA CHANG, Naval Surface Warfare Center Puget Sound Detachment, DAVID DOWLING, Mechanical Engineering, University of Michigan — Acoustic measurements in wind tunnels are notoriously difficult because of machinery noise from the tunnel, and aerodynamic noise generated within the tunnel and by the test model; and because the potentially-weak signals of interest may share a common frequency range with these noise sources. Thus, when model changes are made, localizing any new aeroacoustic source(s) and determining their level(s) is challenging. This presentation provides such experimental results for the aeroacoustic source associated with addition of a 6 mm gap in a 0.5-m-by-1.0-m flat plate aligned with the flow direction. When present, the gap was located 0.40-m from the plate’s leading edge. The measurements were collected in the Anechoic Flow Facility of the Naval Surface Warfare Center – Carderock Division at nominal air speeds of 19 and 30 m/s using a 0.6-m-diameter spiral microphone array with 24 elements placed 1.24 m from the plate and separated from the air flow by a thin barrier. Measurements in the 4-to-12 kHz frequency range were processed using conventional and high-resolution beamforming methods with and without noise-reference subtraction. Source location and level estimation for the gap-induced aeroacoustic sound source was successful at nominal signal-to-noise ratios of ~20 dB.

1Supported by NAVSEA through the NEEC.

10:44AM S06.00002 Mixed-porosity airfoil acoustics. AREN HELLUM, NUWC-Newport — Porous materials have been shown to reduce the noise produced by lifting surfaces. Acoustic and wake velocity measurements have been made on several arrangements of porous material including full-chord impermeability, full-chord porosity, and multiple chordwise variations of mixed porosity. The “mixed” porosity arrangements are produced by changing the area fraction constructed of porous material. Two different porous materials have been employed. The results are made nondimensional and compared to published data. These comparisons indicate that the noise reduction associated with poroelastic foils is associated with the porosity rather than the elasticity of the material. A percolation-based physical model to explain elevated noise production at high frequencies is also proposed.

10:57AM S06.00003 Approximate Solutions for Nonlinear Acoustic Pulses Propagating Upstream in a Subsonic Flow. FATEMEH BAHMANI, Chalmers University of Technology, MARK CRAMER, Virginia Polytechnic Institute and State University, TOMAS GRONSTEDT, Chalmers University of Technology — We have studied acoustic pulses traveling upstream in one dimensional subsonic flow. A sinusoidal pressure pulse is imposed at the right boundary and the transient wave propagation is studied. The undisturbed flow is assumed to be inviscid with uniform density and entropy. The flow velocity is taken to be uniform. N-waves are observed to form after the shock formation time. Due to interactions of the shocks with the waves in front and behind it, the wave amplitude decreases and the wavelength increases as the pulse wave propagates upstream. The variation of pressure coefficient with time and the strength of the pressure disturbances are presented. This solution provides a rough estimation of upstream traveling pulses and can be used to guide and check computations in many practical fluid mechanics applications.

11:10AM S06.00004 Reduced-Order Inversion of Volumetric PIV for Noise Source Characterization. ADAM NICKELS, JEFF HARRIS, Applied Research Laboratory Pennsylvania State University, ALEXANDER MYCHKOVSKY, JAMES WISWALL, KRISTIN CODY, Naval Nuclear Laboratory, TED BAGWELL, Applied Research Laboratory Pennsylvania State University — Flow induced noise sources are often highly three-dimensional, turbulent phenomenon that require knowledge of the three-dimensional velocity-gradient tensor over significant spatial and temporal domains to fully characterize. To address these needs, volumetric-PIV is used to measure the cross-section of a turbulent jet, providing direct measurements of the time-dependent velocity-gradient tensor. Synchronously obtained acoustic pressure measurements allow for correlation of near field quantities with acoustic features of the flow. To better elucidate the spatially and temporally coherent flow features related to the acoustic source, spectral-POD is applied to the velocity field and correlated with the acoustic pressure.

11:23AM S06.00005 Flow-based decomposition of turbulent-jet noise. MARCO RAIOLO, University Carlos III of Madrid, DANIELE RAGNI, TU Delft — Turbulent jet noise is a complex phenomenon characterized by the interaction of modal and non-modal features in the flow field, the former usually defined as wave-packets in literature. In this work the velocity field of a turbulent round jet at Re=30000 is measured using low-repetition rate tomographic Particle Image Velocimetry. Features in the flow field are extracted on a statistical basis using Proper Orthogonal Decomposition, revealing both non-modal features and features with a clear modal behavior in the streamwise direction which are compatible with the wave-packets reported in literature. The temporal evolution of both velocity and pressure of these features, not readily available from the measurements, is estimated through the Galerkin projection of advection equation and of the Euler pressure equation. This approach is advantageous since it also provides estimates of the interaction between several features/modes. Finally, a similar Galerkin projection is attempted on the Lighthill’s equation in order to provide a flow-based decomposition of the far-field noise produced by the jet and to estimate the noise emission due to the cross-interaction of the flow-field modes.

11:36AM S06.00006 Lagrangian Analysis of Intermittent Sound Sources in a Flow-through Square Duct Containing Two Circular Orifice Plates. VINEETH NAIR, C.P. PREMCHAND, Indian Institute of Technology Bombay, Mumbai - 400076, K.V. REEJA, P.R. MIDHUN, MANIKANDAN RAGHUNATHAN, RAMAN I. SUJITH, Indian Institute of Technology Madras, Chennai - 600036 — Tonal sound production in pipe flow through orifices is a manifestation of self-sustained pressure oscillations that occur when positive feedback is established between an acoustic mode of the pipe and vortex shedding. These oscillations, which can be established in segmented solid rocket motors, gas transport pipelines, and automobile exhaust systems are undesirable as they result in structural damage due to fatigue failure. The onset of such oscillations (instability) in turbulent flows for changes in operating conditions from stable operation happens via an intermittent regime comprising bursts of pressure oscillations that are governed by slow (hydrodynamic) and fast (acoustic) time scales. In this study, we focus on the establishment of self-sustained oscillations in the flow through a square pipe containing two circular orifice plates by varying the mass flow rate through the pipe. We extract the coherent structures responsible for sound production at the acoustic time scale from measured time-resolved velocity fields using dynamic mode decomposition (DMD) and the framework of Lagrangian Coherent Structures (LCS). We find that there are noticeable similarities in the sound-producing coherent structures obtained during the bursting stage of intermittency and instability.

1We are thankful to IRCC - IIT Bombay for their financial support (Grant No. 16IRCCSG006).
10:31AM S07.00001 Representative subsampling of sedimenting blood1. HOWARD STONE, Princeton University, BHARGAV RALLABANDI, UC Riverside, JANINE NUNES, ANTONIO PERAZZO, Princeton University, SERGEY GERTHEIN, Abbott Point of Care — It is often necessary to extract a small amount of a suspension, such as blood, from a larger sample of the same material for the purposes of diagnostics, testing or imaging. A practical challenge is that blood sediments noticeably on the time scale of a few minutes, making a representative subsampling of the original sample challenging. Guided by experimental data, we develop a Kynch sedimentation model to discuss design considerations that ensure a representative subsampling of blood, from a container of constant cross-sectional area, for the entire range of physiologically relevant hematocrit over a specified time of interest. Additionally, we show that this design may be modified to exploit the sedimentation and perform the subsampling to achieve either higher or lower hematocrit relative to that of the original sample. Thus, our method provides a simple tool to either concentrate or dilute small quantities of blood or other sedimenting suspensions.

1We thank Abbott Point of Care Inc. and the NSF, CBET-1702693, for partial support of this work.

10:44AM S07.00002 Rearrangement Dynamics in Dense Particulate Systems1. DOUGLAS DURIAN2, University of Pennsylvania — Together at Penn, Jerry Gollub and I jointly supervised the PhD theses of two superlative graduate students as part of Penns MRSEC. Kerstin Nordstrom (assistant professor, Mount Holyoke) and Jennifer Rieser (postdoc, Georgia Tech). In this talk I will review Kerstins work on dynamical heterogeneities in a compressed suspension of hydrogel particles under shear. In particular, flow is mediated by intermittent rearrangements that diverge in size as a universal power law of the local relaxation time, i.e. the reciprocal of the pitter-pattering sound of raindrops on lakes and pools to the thunderous rumble of falls plunging into the rivers flowing below, these sounds can elucidate the mechanisms and evolution behind their origins. Particularly, solid objects can create different sounds when they impact the water surface water depending on their size, shape, speed, etc. Herein, we study these sounds by driving hydrogelic spheres into a quiescent tank and capture the accompanying sounds with in-air microphones and sub-surface hydrophones. Spheres (diameter: 10 mm-24 mm) impacting water at low to high velocities (1-6 m/s) were tested, revealing that the specific sealing phenomena of the cavities have distinct acoustic signatures. Synchronizing high-speed images of these impacts with audio signals captured by acoustic sensors reveal interesting and unique distinct sounds created by the unique cavity sizes, bubble shedding patterns and in some cases the rebounding Worthington jets. These observations help tie the water entry hydrodynamics and acoustics together in a definitive way.

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S07 Non-Linear Dynamics and Chaos IV 211 - Patrick Tabeling, ESPCI Paris

10:31AM S07.00003 Disentangling Resolution, Precision, and Inherent Stochasticity in Fluid Mixing, NICHOLAS GUELLETTE, LEI FANG, Stanford University, SANJEEVA BALASURIYA, University of Adelaide — Reliable measurement, simulation, and analysis of dynamical systems rely on appropriately bounded uncertainty. Errors that lead to uncertainty naturally arise from finite precision or resolution, but an additional unappreciated source of uncertainty is the effective stochasticity associated with nonlinear dynamics. Here we describe and quantify the interplay between these three sources of uncertainty using a recently developed framework known as stochastic sensitivity theory. Using fluid mixing as a test case and considering data from an analytical flow, a laboratory experiment, and geophysical observations, we show how to delimit regimes that are limited by finite resolution or by inherent stochasticity. We arrive at the surprising conclusion that in some cases, refining the resolution of a measurement or simulation can actually be counterproductive and lead to an outcome that is less faithful to the true dynamics.

10:45AM S07.00004 Nonlinear Granular Electrostatics, TROY SHINBROT, Rutgers University — Anyone who has run a sandblasting machine can attest that flowing sand generates a robust and painful stream of electric shocks to the operator. The same mechanism that produces these shocks also generates lightning in sandstorms, and influences the formation of asteroids as well as industrial systems such as fluidized beds. We describe recent progress showing that multiple length and time scales are involved in granular charging, which produces surprising effects. These include distinct charge patterns and oppositely charged surfaces, and competing charging and discharging times that can predictably lead to decreased electrification with increased frequency of vibration of granular beds. None of this work is of the calibre of Jerry Gollub’s, but I present it as an homage: work that I think he would have enjoyed.

10:57AM S07.00005 Chaos and disorder in microfluidics1. PATRICK TABELING, ESPCI — Dynamics obviously plays an important role in microfluidics: how long does it take for a biochemical reaction to develop, how long does it take for a microdroplet to form... More specifically, chaotic dynamics plays an important role in the domain of micromixing. Devices based on chaotic mixing, designed to accelerate mixing, are now currently used in the laboratories. To celebrate the memory of Jerry, I will show a few examples where chaos and disorder, in the way he liked to think about, are exploited to develop certain functionalities in microfluidic devices.

1ESPCL, IPGG, CNRS, Horizon2020
where the internal deformation radius is a maximum. That is partially stabilized by viscosity. Calculations of the growth rate at several latitudes indicate that the instability is enhanced in the Tropics. Despite a highly idealized formulation that assumes a purely meridional basic state and makes a local f-plane approximation, the stability analysis successfully predicts a length scale of the disturbance, a latitude for its origin, and a critical Reynolds number that agree well with accompanying numerical results. Realistic western boundary current profiles undergo a horizontal shear instability that is partially stabilized by viscosity. Calculations of the growth rate at several latitudes indicate that the instability is enhanced in the Tropics where the internal deformation radius is a maximum.

Institution — A linear stability analysis of the shallow-water system in the tropical ocean examines the stability of the western boundary current and its latitudinal dependence. Despite a highly idealized formulation that assumes a purely meridional basic state and makes a local f-plane approximation, the stability analysis successfully predicts a length scale of the disturbance, a latitude for its origin, and a critical Reynolds number that agree well with accompanying numerical results. Realistic western boundary current profiles undergo a horizontal shear instability that is partially stabilized by viscosity. Calculations of the growth rate at several latitudes indicate that the instability is enhanced in the Tropics where the internal deformation radius is a maximum.

12:02PM S07.00008 Watercolors - Measuring the light field in natural waters, NICHOLAS TUFILLARO, dynamic penguin, llc. --- Visible light is one the few wavelengths that has any significant transmissivity through natural waters: oceans, lakes, streams, and ponds. Light is a primary tool for investigating biological or physical processes in water from sensors in orbit, instruments on ships, or spectrometers in the water itself. I will focus on describing how to see — in vivo — the activities of phytoplankton in natural waters, both individual micro-swimmers (~10 microns) up to large assemblages of diatoms (~10 km), and how these measurements are used to understand and gauge biological productivity at both the watershed and global scale. The talk will also discuss a few challenges in trying to do precision measurements in natural environments, instead of the our more natural habitat, the 'lab,' and some of the changes that are helpful to optical instrument designs, measurement procedures, and data analysis.

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S08 Flow Instability: General I 212 - Bud Homsy, University of British Columbia

10:31AM S08.00001 Investigation on Chemical Garden Pattern in Hele-Shaw cell by Interfacial Rheology, TARO MAEDA, YUICHIRO NAGATSU, Department of chemical engineering, Tokyo University of Agriculture and Technology — Chemical garden is an experiment with a precipitation reaction. Precipitation is formed by placing a precipitating salt such as cobalt chloride in a aqueous solution of sodium silicate. In recent years there have been several reports on pseudo 2D chemical garden experiments in Hele-Shaw cell. We also experimentally investigate the influence of concentration of CoCl₂ on chemical garden patterns in Hele-Shaw cell. As the displaced fluid, 3.13 M sodium silicate solution is used, while 0.10~6.25 M cobalt chloride solution is used for injection liquid. Filament pattern is observed at high concentration, spiral pattern is confirmed at middle concentration and flower pattern is emerged at low concentration. These results show that the change in concentration affects the chemical reaction rates and the precipitation rates. Therefore, we analyze such influences by performing some interfacial rheological experiments. We find that the result of interfacial LAOS (large amplitude oscillatory shear) measurement is linked to the selection of formed pattern.

10:44AM S08.00002 Geometrically-weighted Modal Analysis Technique, KOUROSH SHOELE, TSOKANG WANG, Florida State University — Modal decomposition techniques have been used to analyze complex flow and explore how they can be explained with a low-dimensional model. The data-driven methods such as proper orthogonal decomposition (POD) and dynamic mode decomposition (DMD) are used to extract coherent structures in the form of spatial modes. However, the classical data-driven modal decomposition methods have no spatial recognition and are not applicable to time-dependent grids which is common in shape-changing applications. We propose a novel method, the geometrically-weighted modal analysis, utilizing differential geometry mapping techniques to solve this issue. The deforming geometries are transformed into fixed domain based on their geometric characteristics and with the use of divergence-free mapping then the modal analysis is applied. Through different examples, we demonstrate the capability of this method to accurately capture the flow feature and dynamics of complex fluid-structure interaction systems.

10:57AM S08.00003 Experimental Study of Roll-Hydrothermal Wave Coexistence in Convection Driven by Buoyancy and Thermocapillarity, MICHAEL SCHATZ, BRETT TREHONG, Georgia Institute of Technology, JOSHUA BARNETT, Stanford University, MINAMI YODA, ROMAN GRIGORIEV, Georgia Institute of Technology — Buoyancy-thermocapillary convective flow in a volatile fluid with a free surface and a horizontal temperature gradient arises in a variety of situations. Previous work examining buoyancy-thermocapillary flow in a rectangular geometry showed that hydrothermal waves are usually found in the limit where the dynamic Bond number approaches zero while a stationary convection roll pattern is found in the limit where the dynamic Bond number is of order unity while the Marangoni number is held as a control parameter. Linear stability analysis predicts a dynamic Bond number regime in between these limits in which static convection rolls coexist in the domain with hydrothermal waves that propagate along the direction of the temperature gradient [Grigoriev and Qin, JFM 838, 248 (2018)]. By probing this regime over a range of dynamic Bond numbers, we examined these predicted dynamics using an experimental cell containing silicone oil imaged with a shadowgraph technique.

This work was supported by the National Science Foundation under Grant No. CMMI-1511470.
11:10AM S08.00004 Stability analysis of supersonic round jet with temperature non-uniformity, MONIKA CHAUHAN, K. TODD LOWE, LUCA MASSA, Virginia Tech — We perform parallel and parabolized stability analyses of round supersonic jets supported by the adiabatic expansion in turbulent nozzle. We carry out Reynolds-average Navier-Stokes calculations in two and three dimensions and test various turbulence models to resolve the effects of incoherent turbulence created by the nozzle walls on coherent structures in the shear layer. The computed mean profiles are in reasonable agreement with the experiments using the SST turbulence model and show the presence of a region of low momentum near the axis due to matching of Mach number in the cold and hot streams. After validating the mean profiles against the experimental data, we determine the stability characteristics with axisymmetric injection at the upstream of the throat. We find that the maximum axisymmetric mode is weakened by cold injection at the axis, while the first and second circumferential modes are of similar magnitude but exist for a reduced range of Strouhal numbers. Next, we evaluate the effect of circumferential temperature non-uniformity in the mean profile by calculating the stability of three-dimensional mean-profiles with resonant coupling against the circumferential instability modes.

11:23AM S08.00005 Centrifugal instabilities in curved free shear layers: direct computations in the nonlinear regime, OMAR ES-SAHLI, ADRIAN SESCU, Mississippi State University, USA, MOHAMMED APSAR, University of Stratchclyde,UK — Curved free shear layers abound in many engineering applications involving complex geometries, such as backward facing step flows, wall injection, the flow inside side-dump combustors, or flows around vertical axis wind turbines. Most of the previous studies involving centrifugal instabilities have been focused on wall-bound flows, where the so-called Taylor vortices in enclosed geometries or Görtler vortices in boundary layer flows on concave surfaces are generated. Centrifugal instabilities in curved free shear layers, however, did not receive sufficient attention partly because these flows are mostly dominated by Kelvin-Helmholtz instabilities. Under certain conditions, however, longitudinal instabilities in the form of Görtler vortices can occur, which – alone or in combination with Kelvin-Helmholtz type instabilities – may be susceptible to secondary instabilities and ultimately to turbulence. We study the development and growth of nonlinear Görtler vortices evolving inside curved free shear layers in both incompressible and compressible regimes, using direct numerical solution to the Navier-Stokes equations. Results for different flow conditions are reported, along with discussions of challenges associated with simulating these types of flows.

11:36AM S08.00006 Kelvin-Helmholtz shear instability with strong thermal nonequilibrium1, MYOUNGKYU LEE, MICHAEL A. GALLIS, JACQUELINE H. CHEN, Sandia National Laboratories — The effect of strong thermal nonequilibrium on the Kelvin-Helmholtz (K-H) instability is studied with the Direct Simulation Monte Carlo (DSMC) method. Transport properties of gases vary with temperature and, in general, are a function of a single temperature, which assumes the translational, rotational, and vibrational energies are locally in equilibrium at macroscopic scales. The local equilibrium assumption is invalid when flows with K-H instability undergo certain conditions whereby the residence time is short compared to thermal relaxation time scales, for example, in reactive turbulent jet flows at high Mach number with strong temperature and/or velocity gradients. The DSMC method models the fluid flows using molecule-simulators, following the probability density functions for different energy modes. First, we demonstrate the validity of the DSMC method by simulating helium-argon flows undergoing K-H instability with DSMC and compare the result with solutions from continuum direct numerical simulation. Next, hydrogen and air are used as working fluids to introduce a strong thermal gradient and nonequilibrium. Finally, we analyze the energy transfer between length-scales as a function of the nonequilibrium ratio, e.g., \[ \phi_{\text{gb}} = \frac{3T_{\text{ vib}} (T_r + T_{\text{rot}} + T_{\text{tr}})}{T_{\text{rot}} + T_{\text{tr}} + T_{\text{tot}}} \].

1This research was supported by the US DoE, Office of Basic Energy Sciences, and used resources of the National Energy Research Scientific Computing Center, a US DoE Office of Science User Facility (Contract No. DE-AC02-05CH11231).

11:49AM S08.00007 Direct Numerical Simulations of the Span-wise Asymmetric Kelvin-Helmoltz Instability, SCOTT WIELAND, DAVID FRITTS, THOMAS LUND, GATS Inc. — Recent advancements in computational power have allowed the study of more complex fluid interactions, including exploring previously untouched regimes of the Kelvin-Helmholtz instability (KHI). Theory has predicted interesting effects stemming from the introduction of span-wise variations to the perturbations and shear layer of KHI. To investigate this, direct numerical simulations have been carried out using the Complex Geometry Compressible Atmospheric Model. These simulations have been performed at a Reynolds number of 5000 with a Richardson number of 0.05 and have explored both the effects of span-wise asymmetries in the shear layer thickness and the perturbations applied as initial conditions. The results obtained confirm the predicted development of the interactions between misaligned billows as characteristic x-shaped “knots.” These knots, though, produce intense events at faster time scales expressed as vorticity and energy dissipation at least an order of magnitude higher than the standard KHI developing in the background. These results will also be compared to observations of similar KHI events obtained from imaging of the polar mesospheric cloud layer and from measurements taken from the Andes LIDAR Observatory.

12:02PM S08.00008 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2019 10:31AM - 12:15PM –
Session S09 Industrial Applications: General 213 - Dana Greco, University of British Columbia

10:31AM S09.00001 Aerodynamic Performance of a Rim Driven Thruster, MAXWELL KOGLER, CONOR PACE, HASAN RAZA, MICHAEL VU, OLEG GOUSHCHA, Manhattan College — Rim driven thruster consists of a fan actuated into a rotational motion by a mechanism located at its outer radius, rather than by a centerline shaft as seen in conventional fan designs. Such configuration eliminates a need for a centerline hub containing driving mechanism, allowing for an undisturbed core flow of fluid through the centerline region of the fan. In this study we present a rim driven thruster actuated by an interaction between electric and magnetic fields, similar to the brushless motor with entire fan and supporting structure resembling rotor and stator, respectfully. This thruster operates in air and can be used as a propulsion system in airplanes. The aerodynamic performance was evaluated experimentally by measuring upstream and downstream velocity fields and thrust forces at various flight conditions. Results from the experiments are compared to the predictions made by the Blade Element Theory.
10:44AM S09.00002 Effect of surface roughness on numerical modeling of full-scale self-propulsion\(^1\), HENRIK MIKKESEN, JENS HONORE WALTHER, Technical University of Denmark — When predicting the full-scale performance of a vessel, the resistance from surface roughness is important, since it can account for up to 15\% of the total resistance for a newly built vessel. We compare full-scale computational fluid dynamics (CFD) simulations with speed trial measurement from six sister ro-ro vessels. The study includes extensive validation and verification of both resistance, open-water and self-propulsion simulations. Full-scale resistance and open water as well as model scale self-propulsion simulations show good agreement with towing tank measurements and predictions. However, the full-scale self-propulsion simulations using the traditional approach of including the roughness estimated by the empirical formulae of Tounsi significantly underestimates the power from the speed trials. By including the effect of hull and propeller roughness directly into the CFD simulation, by modifying the wall functions, the discrepancy between CFD and speed trail measurements now reach an acceptable level. As a result, full-scale CFD simulations is becoming a viable and accurate alternative to transitional towing tank experiments.

\(^1\)The research was supported by The Danish Maritime Fund under grant 2018-11.

10:57AM S09.00003 Thermal mapping of Hollow Cathodes to model the thermal loads of Iodine Ion Propulsion\(^1\), PAUL WINNER, Embry-Riddle Aeronautical University, Daytona Beach, DR. RICHARD BRANAM COLLABORATION, KIRK BOEHM COLLABORATION — Iodine possesses desirable qualities that could make it a potential fuel of choice in future space missions requiring ion propulsion, potentially replacing Xenon; if the thruster can have similar endurance despite Iodine’s corrosiveness. A computer-aided design model of a hollow cathode with a Lanthanum Hexaboride insert was created in Solidworks, using its thermal load simulator to generate an approximate temperature model of it in operation. Data was then collected to refine the model by getting temperature readings of a hollow cathode in operation by attaching type K thermocouples to the outside and inside of the hollow cathode and firing it in a vacuum chamber. Differences between the thermal model and the experimental data will be discussed in addition to the assumptions made within the thermal model. Based on the data, the hollow cathode is far below expected temperatures, though there are several sources of error to explain this discrepancy.

\(^1\)NSF Grant #1659710

11:10AM S09.00004 Electrohydrodynamics of charge-injecting high-voltage electrodes, XUEWEI ZHANG, Texas A&M University Kingsville — In liquid dielectrics, initiation of electrical breakdown usually takes place at the electrode/liquid interface under a much lower electric field than required by molecular ionization, even if the material is highly purified. One of the main causes is the fact that there exist micro-protrusions and micro-voids on the electrode surface. Small use of charge injection to improve electrical breakdown strength has been demonstrated in previous works. The objective of this paper is to propose and numerically study a new design of charge injecting electrodes inspired by inkjet printer. In a needle-plane configuration, the needle electrode is hollow and has microchannels connecting its interior to the outside, both filled with the same dielectric liquid. The highly inhomogeneous electric field has electrostriction effect, creating a low-pressure region at the needle tip. The pressure difference across the microchannels will drive liquid flow from inside the electrode to the outside. During this, the liquid jets will be electrified and carrying the charge with the same sign as the electrode (homocharge). We build the electrohydrodynamic model and develop a numerical program to simulate this process. The steady-state simulation results confirm that the hollow electrode with microchannels can effectively inject homocharges which enhance the electrical conductivity and lower the electric field in the region near the electrode surface.

11:23AM S09.00005 Modelling Purification of Flue Gas in Porous Catalytic Media\(^1\), KRISTIAN KIRADJIEV, CHRISBreward, IAN GRIFFITHS, University of Oxford, DONALD SCHWENDEMAN, RPI, UWE BEUSCHER, VASUDEVAN VENKATESHWARAN, W. L. Gore and Associates — In this talk, we present a mathematical model for flue-gas purification in a porous filter with catalyst. In particular, we consider a device that converts gaseous sulphur dioxide into liquid sulphuric acid which accumulates, causing clogging. Using the theory of homogenisation, we develop a multiscale model that takes into account local properties of the filter to describe the overall device operation. We explore the effect of changing various dimensionless parameters on the filter efficiency. We also consider asymptotic reductions to the full system and compare them with full numerical solutions.

\(^1\)Supported by EPSRC Centre For Doctoral Training in Industrially Focused Mathematical Modelling (EP/L015803/1) in collaboration with W. L. Gore and Associates.

11:36AM S09.00006 Enhancement of convective cooling in solar photovoltaics, MARC CALAF, BROOKE STANISLAWSKI, TODD HARMAN, University of Utah, RAUL B. CAL, Portland State University — At present, most research on solar photovoltaics (PV) is focused on improving cell efficiency at 25°C, while most world-record efficiencies have been measured, and which neglects the fact that in outdoor conditions the air temperature grows significantly higher. Reduction of undesirable thermal effects can be achieved by either decreasing heat generation at the cell level or by increasing heat dissipation. Here, heat dissipation is exploited through enhanced convection, which has remained fairly unexplored. It is now well-accepted that solar module temperature increases lead to undesired power losses. The rate of efficiency decrease per degree of temperature rise is quantified by the “temperature coefficient.” For a silicon cell, the efficiency of the cell drops by about 0.4% with the increase of every one degree Celsius above 25°C, and solar modules in real atmospheric conditions can typically operate at upwards of 25°C above the ambient temperature. While ongoing efforts continue to reduce the thermal loadings and manipulation the operating temperatures of solar PV modules is sought out within large-scale solar farms by developing new solar farm arrangements that boost the convective heat transfer between the modules and the atmospheric flow. For this purpose, large-eddy simulations of realistic solar PV farms are conducted, where results will illustrate whether there exists any preferential solar farm arrangement for enhanced cooling and hence increased solar PV efficiency.

11:49AM S09.00007 Effect of moving ground and rotating wheels on the flow over a high speed passenger train, MOHAMMAD ASIF SULTAN, DIBYENDU KONAR, Research Scholar, Indian Institute of Technology Kharagpur, SUBHRANU ROY, Professor, Indian Institute of Technology Kharagpur — High speed passenger trains are gradually coming up successfully as a mode of modern transportation. The impact of moving ground and rotating wheels on the flow around a high speed train has been numerically investigated. A Reynolds number of 1.85x10\(^6\) based on the air flow velocity and height of the train is used in the CFD modelling using k-\(\varepsilon\) turbulence model. The total aerodynamic drag were calculated for the three cases: stationary ground, moving ground, moving ground with rotating wheels. Moving ground boundary condition is found to eliminate the boundary layer near the ground which affects the flow beneath the train and changes the pressure distribution. The rotating wheel boundary condition tends to increase the velocity of flow in the bogie region. With moving ground, there is 10.2% increment in the total drag which further increases by 1% when rotating wheels are considered. The simulations show that bogies also have a major contribution on the total drag. It is around 15.2% with stationary ground, increases to 22.8% while ground is moving and finally attains 25.1% when the most realistic case of both moving ground and wheel rotation is considered.
coefficients computed via RANS and the experiments. Reynolds number varying between $Re \leq 5 	imes 10^5$, where $L$ is the streamwise length of the bump. The results obtained by solving the Reynolds-averaged Navier-Stokes (RANS) equations using the Spalart-Allmaras model. The new bump geometry is defined by a Gaussian profile along the span of the cone. The continuous variation of the cone diameter regulate the changes in the shedding frequencies and in some cases controversial. In particular, in LES performed by different groups it was found that increasing grid resolution or decreasing subgrid-scale dissipation leads to a deterioration of the agreement with the experiments with a too short mean recirculation region on the cylinder side. We show that this paradox can be explained by the fact that the upstream corners in the numerical simulations are perfectly sharp while they have a certain degree of roundness in experiments. Indeed, highly-resolved LES are presented showing that even very small values of the corner curvature radius have a dramatic impact on the numerical solution on the cylinder lateral sides. Sensitivity studies carried out by the BARC contributors were not conclusive in detail to characterize the flow over a cone. The continuous variation of the cone diameter regulate the changes in the shedding frequencies along the span of the cone.

10:44AM S10.00002 Vortex formation at the apex of an oblique cone in uniform cross-stream, AL SHAHRIAR, KOUROSH SHOELE, RAJAN KUMAR, Florida State University — The present investigation is concentrating on the dynamics of the vortex in the wake region of the cone for high angle of attacks. As the flow built up from the tip toward the base, it is associated with the swept boundary layer, 3D coherent and turbulent structures, asymmetric vortices and no fixed length scale. Based on the base diameter, the study was conducted for moderate to higher Reynolds number with LES and a wall function. Semi-body conformal grid has been generated for higher accuracy and resolution near the body. The tip of the cone has been found very sensitive to the coherent structures formed downstream of the flow. The instantaneous frequencies, wavelengths, turbulent structure, and shedding angles have analyzed and in some cases controversial. In particular, in LES performed by different groups it was found that increasing grid resolution or decreasing subgrid-scale dissipation leads to a deterioration of the agreement with the experiments with a too short mean recirculation region on the cylinder side. We show that this paradox can be explained by the fact that the upstream corners in the numerical simulations are perfectly sharp while they have a certain degree of roundness in experiments. Indeed, highly-resolved LES are presented showing that even very small values of the corner curvature radius have a dramatic impact on the numerical solution on the cylinder sides leading to a very good agreement with experimental data.

10:57AM S10.00003 Effect of corner rounding on the flow around a 5:1 rectangular cylinder. A paradox explained?, BENEDETTO ROCCHIO, ALESSANDRO MARIOTTI, MARIA VITTORIA SALVETTI, DIC, University of Pisa — The high Reynolds number flow around a rectangular cylinder, having chord-to-depth ratio equal to 5, is the object of the benchmark BARC, collecting several experimental and numerical results. This configuration is characterized, in particular, by flow separation at the upstream corners and reattachment on the cylinder side. A large dispersion was observed in the numerical predictions of the flow features and quantities on the cylinder lateral sides. Sensitivity studies carried out by the BARC contributors were not conclusive and in some cases controversial. In particular, in LES performed by different groups it was found that increasing grid resolution or decreasing subgrid-scale dissipation leads to a deterioration of the agreement with the experiments with a too short mean recirculation region on the cylinder side. We show that this paradox can be explained by the fact that the upstream corners in the numerical simulations are perfectly sharp while they have a certain degree of roundness in experiments. Indeed, highly-resolved LES are presented showing that even very small values of the corner curvature radius have a dramatic impact on the numerical solution on the cylinder sides leading to a very good agreement with experimental data.

11:10AM S10.00004 RANS simulations of a turbulent separated flow validation test-case., MADALINE SAMUELL, OWEN WILLIAMS, ANTONINO FERRANTE, University of Washington, Seattle — A new validation bump-geometry for turbulent separation has been analyzed experimentally, in our $3' \times 3'$ wind-tunnel, and computationally, by solving the Reynolds-averaged Navier-Stokes (RANS) equations using the Spalart-Allmaras model. The new bump geometry is defined by a Gaussian profile in the streamwise direction and an error function in the spanwise direction, such that the turbulent flow over the bump separates, under the adverse pressure gradient, in the downstream region of the bump. Sensitivities of skin-friction and pressure coefficients, as well as the extent of separation to various geometric and flow parameters (e.g., bump height to length ratio, distance of the bump from the top-wall to length ratio, boundary layer thickness of incoming flow on bottom and top walls) have been analyzed. The study has been performed for a range of Reynolds number varying between $0.59 \times 10^6 \leq Re_L \leq 3.5 \times 10^6$, where $L$ is the streamwise length of the bump. The results obtained by using different turbulence models have also been compared. Initial two-dimensional RANS simulations show discrepancies between the pressure coefficients computed via RANS and the experiments.

11:23AM S10.00005 Wall-modeled LES of flow around a prolate spheroid at various angles of attack., XINYI HUANG, XIANG YANG, Pennsylvania State University — We conduct wall-modeled large-eddy simulations (WMLES) of flow around a 6:1 prolate spheroid at various angles of attack (AOA) from $10^\circ$ to $30^\circ$. An equilibrium wall model is used with a coarse mesh deploying 10 to 20 cells within one local boundary layer thickness. The Reynolds number based on the long axis length and the freestream velocity is $4.2 \times 10^6$. This flow was extensively examined experimentally by Simpson et al., in the 90s. We examine the pressure coefficients at a few streamwise locations in the azimuthal direction, as well as the locations of flow separation. Our WMLES results agree reasonably well with the previous measurements, and regions of secondary separation are also captured. WMLES provide more detailed flow information than Reynolds Averaged Navier Stokes. In our simulations, a pair of counter-rotating vortices is reproduced. These vortices originate from the leeward surface and extends in the streamwise direction, leading to notable downwash. The resulting wake is non-axisymmetric. Skewness of the wake is measured and compared to predictions of a lift line model.
We gratefully acknowledge funding by the Air Force Office of Scientific Research under FA9550-16-1-0392. Mattia Serra would like to acknowledge support from the Schmidt Science Fellowship.

1Supported by USDA-NIFA SCRI Award 2016-51181-25403. Computations performed using USDA-ARS SCINet.

2Also, Department of Computer Science

11:49AM S10.00007 Lagrangian Flow Separation in External Aerodynamics

—

KLOSE, GUSTAAF JACOBS, San Diego State University, MATTIA SERRA, Harvard University — Kinematic aspects of flow separation in external aerodynamics are investigated by revealing the initial motion of upwelling fluid material from the wall and its relation to the long-term attracting manifolds in the flow field. With direct numerical simulations of a circular cylinder and a cambered NACA 65(1)-412 airfoil, the location of initial fluid upwelling, the so-called spiking point, is determined from the curvature of advected material lines and from high-order numerical derivatives of the wall-normal velocity. While the short-time kinematics are governed by the formation of a material spike upstream of the zero-skin-friction point, over longer times the trajectories of the fluid tracers are guided by attracting ridges in the finite-time Lyapunov exponents once they leave the vicinity of the wall. The combination of initial fluid upwelling, asymptotic separation line, and attracting Lagrangian Coherent Structures draws a comprehensive picture of the mechanics of flow separation in external aerodynamics.

11:36AM S10.00006 High Schmidt Number Washout of Sodium Hypochlorite or a Viscosifying Solute Shielded by Topography

—

MIN, PAUL F. FISCHER, ARNE J. PEARLSTEIN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — In many cleaning applications, including washing of fresh-cut produce, surface topography shields soluble contaminants from “washout.” Here, we report computations of washout by two-dimensional flow downstream of a backward-facing step, with a contaminant initially confined to a square domain just downstream of the step, with edge length equal to the step height. We consider three cases. In the first, we focus on removal of NaClO for Reynolds numbers (Re) of 10 and 100 (based on step height) at a Schmidt number (Sc) of 2650. For Re = 100, there is a time in the washout process when the maximum concentration shifts from the relatively inaccessible bottom corner behind the step, to a point in the interior of the recirculation zone downstream of the step. The size of the recirculation zone affects washout efficiency, corresponding to a strong Re dependence on Re. In the second case, we model washout of organic exudate that leaks from freshly cut produce, using data for the viscosifying sugar acid sodium gluconate (NaG) at Re = 100 and an infinite-dilution Sc of 4165, accounting for the dependence of viscosity on concentration. Finally, we show results for the case in which NaClO is present in the free-stream, and reacts with NaG downstream of the step.

We gratefully acknowledge funding by the Air Force Office of Scientific Research under FA9550-16-1-0392. Mattia Serra would like to acknowledge support from the Schmidt Science Fellowship.

11:00AM S10.00005 Session S10 Experimental Techniques: Quantitative Flow Visualization III

—

Truscott, Utah State University

10:31AM S11.00001 ABSTRACT WITHDRAWN —

10:44AM S11.00002 Towards quantification of biologically generated turbulence through 3D scanning particle image velocimetry

1

FU, Stanford University, ISABEL HOUGHTON, The Data Institute, University of San Francisco, JOHN DABIRI, Caltech — The role of biologically generated turbulence in scalar transport and ocean mixing remains inadequately understood. Though the turbulent scales created by a single swimmer might be limited to those of the individual organism, recent work has suggested that the larger sizes associated with the aggregates of vertically migrating swimmers can introduce mixing scales relevant to the surrounding water column. Quantifying this process requires that the entire flow field and the full range of mixing scales, from the size of the aggregate to below that of the individual animal, be resolved. While there have been significant advancements in volumetric velocity measurements, the spatial resolution associated with these techniques is typically insufficient for exploring common species of vertically migrating swimmers. Here, we present a scanning particle image velocimetry apparatus for quantifying three-dimensional configurations of vertically migrating swimmers and their volumetric, three-component velocity fields and demonstrate its use on the vertical migrations of brine shrimp Artemia salina.

1U.S. National Science Foundation (grant 1510607)

10:57AM S11.00003 Simultaneous measurements of large-scale flow dynamics and small-scale turbulent properties by a hybrid method combining cross-correlation and optical-flow schemes

1

TIAN, ZHEN-YUAN GAO, SHI-DI HUANG, Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, TIANSHU LIU, Department of Mechanical and Aerospace Engineering, Western Michigan University — Particle image velocimetry (PIV) has been established a standard technique based on cross-correlation scheme for velocity field measurement. While PIV holds advantages in the measurements of large-scale flow dynamics, its accuracy in measuring small-scale turbulent quantities is always unsatisfactory due to limited spatial resolution. On the other hand, optical flow method is born with the capability of extracting high-resolution velocity field up to one vector per pixel [J. Fluid Mech. (2008), vol. 614, pp. 253–291]. In this work, we utilize a hybrid method, combining cross-correlation and optical flow schemes, to analyze the PIV images of global field measured in turbulent Rayleigh-Bénard system. Both the large-scale flow structures and small-scale turbulent properties are obtained simultaneously. It is found that the time-averaged velocity fields are consistent with the results obtained by standard PIV technique. Furthermore, the kinetic energy dissipation and structure functions obtained from present method are compared with those obtained from standard PIV method.

1This work was supported by the National Natural Science Foundation of China under Grant Nos. 11702128 and 91752201.
11:10AM S11.00004 Time-resolved turbulent velocity field reconstruction using a long short-term memory (LSTM). ZHIWEN DENG, Shanghai Jiao Tong University and Pusan National University, YINGZHENG LIU, Shanghai Jiao Tong University, KYUNG CHUN KIM, Pusan National University — This paper focuses on the time-resolved turbulent flow reconstruction from discrete point measurements and non-time-resolved PIV measurements using an artificial intelligence framework based on LSTM. To this end, an LSTM-based proper orthogonal decomposition (POD) model is proposed to establish the relationship between velocity signals and time-varying POD coefficients obtained from non-TR-PIV measurements. An inverted flag flow at Re=6,200 was experimentally measured using TR-PIV for the construction of training and testing datasets and for validation. Two different time-step configurations were employed to investigate the robustness and learning ability of the LSTM-based POD model: a single-time-step structure and a multi-time-step structure. The results demonstrate that the LSTM-based POD model has great potential for time-series reconstruction since it can successfully recover the temporal resolution of POD coefficients with remarkable accuracy, even in high-order POD modes. In addition, a relative error reconstruction analysis was conducted to compare the performance of different time-step configurations further, and the results demonstrated that the POD model with multi-time-step structure provided better reconstruction of the flow fields.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A2B2007117).

11:23AM S11.00005 Extending ”Postage-Stamp PIV” to MHz Rates for Measurement of Turbulent Velocity Spectra. STEVEN BERESH, RUSSELL SPILLERS, MELISSA SOEHNEL, SETH SPITZER, Sandia National Laboratories — Previously, time-resolved particle image velocimetry using a pulse-burst laser demonstrated direct measurement of turbulent velocity spectra at very high frequencies without frozen turbulence assumptions. By accepting reduced laser pulse energy and confining the measurement to a field of view of only 128 x 120 pixels, sequences of 4000 images at 400 kHz were acquired, giving rise to the moniker of ”postage-stamp PIV.” Still, the data reached frequency limitations of approximately 120 kHz due to noise interference. To raise the effective frequency response, the increased speeds of the most recently available cameras obtained data at a sampling rate of 990 kHz, but require overcoming drawbacks in these cameras that can have unfortunate properties for accurate PIV measurements. The increased framing rate oversamples the data and therefore a conventional image-pair correlation can be replaced by multiple-frame image interrogation. The increased accuracy of multiple-frame methods lowers the noise floor and therefore reveals higher-frequency content showing deviation from the theoretical -5/3 power law of the inertial subrange, likely because the turbulent fluctuations have not reached isotropy.

11:36AM S11.00006 Wall Tracking Method for High Resolution Boundary Layer Measurements on a Wing-Fuselage Model. DAVID JEON, California Institute of Technology, CHRISTIAN WILLERT, Deutsches Zentrum für Luft- und Raumfahrt, Kln, DAMIAN HIRSCH, MORTEZA GHARIB, California Institute of Technology — Turbulent boundary layer experiments pose a problem with the large range of length scales that need to be resolved. Measurement techniques tend to focus at either end of this scale. Techniques like holographic PIV primarily measure the region nearest to the wall. Those like conventional PIV typically only resolve down to the log layer. A variation on PIV has been developed by one of us (Willert), where a long-range micro PIV setup is used to resolve down to sub-layer scales. This type of micro PIV setup was used on a wing-fuselage model to measure the boundary layer on the model at Re=2700, from y+=1 in the sub-layer nearly to the outer edge of the boundary layer. In addition, wall shear stress can be computed both using the profile in the sub-layer and the Clauser method through the log-layer and compared against each other. The greatest problem faced with this technique is compensating for apparent wall motion. For example, at the conditions where we can resolve down to y+=1, the apparent wall motion is on the order of 10 wall units. After compensation, this drops below 0.1 wall units. Therefore, the wall tracking method used is critical to get data close to the wall.

1This work was supported by The Boeing Company through grant number CT-BA-GTA-1.

11:49AM S11.00007 Turbulence characteristics of flow through an open cell metal foam. YOUNGWOO KIM, CHANHEE MOON, KYUNG CHUN KIM, Pusan National University — Random nature of stochastic foam provides favourable geometrical properties for thermal applications such as large specific surface area and high porosity. However, notwithstanding intensive investigations for the last two decades, hydrodynamic characteristics of stochastic foam are still poorly understood. Current 3-D printing technology supports printing of transparent complex structures. This study investigates turbulence characteristics of a stochastic foam. Using micro-tomography and stereo-lithography, a transparent stochastic foam was printed. Quantitative flow visualization was performed using refractive index matching technique and time-resolved particle image velocimetry. Mixing and turbulence characteristics were discussed. Mechanical mixing and high turbulence in the stochastic foam is beneficial for thermal applications, but large wake area behind the struts cause high pressure drop. On the basis of the results, a new concept of turbulence due to complex geometry can be named as “structure generated turbulence (S3T)”.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2011-0030013, No. 2018R1A2B2007117).

12:02PM S11.00008 Smartphone PIV. DAVID ARMUJO, LORI CALDWELL, SARBAJIT MUKHERJEE, VLADIMIR KULYUKIN, ANGELA MINICHIELLO, TADD TRUSCOTT, Utah State University — We are developing a smartphone app that performs particle image velocimetry (PIV). This app is called Mobile Instructional Particle Image Velocimetry (mPIV). The intent is to increase the availability of PIV systems to high school and undergraduate students. PIV is used to measure the velocity within a flow field by illuminating neutrally buoyant particles with a laser sheet and recording the motion with a video camera. In mPIV the particles are illuminated with a laser pointer spread into a laser sheet, and the camera is a smartphone. This work is accomplished by integrating the image capture, pre-processing, PIV calculations, and vector output post-processing onto a mobile device. The Java app will be compatible with any smartphone running Android 5.0 Lollipop (API 21) or higher. The system accuracy is benchmarked by comparing a free stream flow field miPIV output with a lab grade system. Results indicate that the system performs well where the illumination quality is high and the flow speeds are below the motion blur limit. Even when these perfect scenarios are not met, students can see flow structures and average velocities that reveal information about the flow field in a way that can inspire them to further study fluid mechanics and its applications.
results also naturally extend to higher dimensional versions of this problem. We maintain a triangulation of the points as they move, while encoding curves as edge weights that enumerate transverse intersections. These output of experiments, this constitutes the problem of topological advection. The current best known topological advection algorithm blends a coherent sets. Since topological entropy is given by the exponential stretching rate of material curves and coherent sets are defined to have problems typically involve either quantifying mixing (e.g. with topological entropy) or finding barriers to mixing (e.g. by identifying Lagrangian vortex. An underlying assumption in past work was that the vortex zone was topologically separated from its exterior. In the present talk, we investigate a series of more challenging examples where this topological condition breaks down. We show that the HLD technique can still describe the topological features of such flows and yield an estimate of the topological entropy.

10:44AM S12.00002 Computation of Topological Entropy in Three Dimensions from Fluid Trajectories1, ERIC ROBERTS, SUZANNE SINDI, University of California, Merced, SPENCER SMITH, Mount Holyoke, KEVIN MITCHELL, University of California, Merced — We introduce and verify an algorithm for estimating a three-dimensional flow’s topological entropy, a measure quantifying the complexity of chaotic dynamics. Analogous to the topological entropy calculation from the “braiding” of system trajectories in two dimensions by Thiffeault, we achieve this in three dimensions by exploiting the collective motion of an ensemble of potentially-sparse system trajectories; as the ensemble evolves in time, the points repeatedly stretch and fold two-dimensional rubber sheets. The topological entropy is bounded below by the exponential growth rate of this sheet, thereby quantifying the flow complexity. New to three-dimensional entropy calculations is the introduction of computational geometry tools: we maintain a Delaunay triangulation of points as they move to record the evolution and growth of a sheet. Because this algorithm requires only trajectory data and no knowledge of governing equations, these results aid greatly in a wide variety of natural and industrial fluid systems, including the large-scale dispersion of pollutants in the Earth’s atmosphere and oceans and the rapidly developing field of microfluidics, all while remaining generally applicable to theorists and experimentalists alike.

1NSF grant number 1808926 titled “Self-mixing Active Fluids.”

10:57AM S12.00003 Topological Advection, SPENCER SMITH, Mount Holyoke College — Fluid advection problems typically involve either quantifying mixing (e.g. with topological entropy) or finding barriers to mixing (e.g. by identifying Lagrangian coherent sets). Since topological entropy is given by the exponential stretching rate of material curves and coherent sets are defined to have boundaries which do not appreciably stretch, both advection problems have a common formulation: find the future state of material curves under the action of the flow. When our knowledge of the fluid system is through sparse data (finite set of trajectories), which is the natural output of experiments, this constitutes the problem of topological advection. The current best known topological advection algorithm blends a braid theory representation of trajectories with a clever coordinate system on the space of closed material curves. I will present a new algorithm which solves the topological advection problem more efficiently and perhaps more naturally. It uses ideas from computational geometry to maintain a triangulation of the points as they move, while encoding curves as edge weights that enumerate transverse intersections. These results also naturally extend to higher dimensional versions of this problem.

11:10AM S12.00004 Topological Helical Vorticity Compression in Ideal Fluids, THOMAS MACHON, University of Bristol — We show how an additional topological conserved quantity arises in ideal fluids whenever the helicity vanishes in such a way that the vorticity field is tangent to a family of surfaces. We give examples of vorticity fields for which this quantity does not vanish, and interpret it as measuring helical compression of vortex lines. We show that if this invariant does not vanish then the flow is not steady, giving a topological obstruction for a vorticity field to come from a steady flow. Finally we discuss relations to the Hamiltonian formulation of the Euler equations.

11:23AM S12.00005 Topological Flow Data Analysis Part 1- Theory and Applications1, TAKASHI SAKAJO, Kyoto University, TOMOKI UDA, Tohoku University, TOMOO YOKOYAMA, Kyoto University of Education — We have investigated a mathematical theory classifying the topological structures of streamline patterns for 2D incompressible (Hamiltonian) vector fields on surfaces such as a plane and a spherical surface, in which a unique combinatorial structure, called partially Cyclically Ordered rooted Tree (COT), and its associated graph (Reeb graph) are assigned to every streamline topology. With the COT representation, one can identify the topological streamline structures without ambiguity and predict the possible transition of streamline patterns with a mathematical rigor. In addition, Uda has recently developed a software converting the values of stream function on structured/non-structured grid points in the plane into the COT representation automatically. It enables us to conduct the classification of streamline topologies for a large amount of flow datasets and the snapshots of time-series of flow evolutions obtained by measurements and numerical simulations, which we call Topological Flow Data Analysis (TFDA). In the three consecutive talks Part1-Part3, we introduce the recent developments of TFDA. In the first part followed by two talks, we will present an overview of basic theory and its applications to atmospheric data.

1This work is supported by Japan Science and Technology.

11:36AM S12.00006 Topological Flow Data Analysis Part 2- Implementation and Software Demonstration1, TOMOKI UDA, Tohoku University, TOMOO YOKOYAMA, Kyoto University of Education, TAKASHI SAKAJO, Kyoto University — Topological data analysis attracts researchers in many fields and it also plays important roles in Topological Flow Data Analysis (TFDA). In the TFDA basis by Sakajo and Yokoyama, streamline patterns are identified by partially Cyclically Ordered rooted Trees (COTs) under topological classification. Since their theory is applicable to a wide range of physical phenomena, there is a rise in demand for an algorithm converting given flow data to COTs. A Reeb graph of a real-valued function, which consists of certain topological sets, is a central key to the conversion. The author has proposed a new algorithm so that the Reeb graph with certain mathematical properties are obtained. Because our formulation is based on persistent homology in topological data analysis, a stability is ensured in our algorithm. We also establish a consistency theorem that bridges a gap between discrete data and continuous data. Furthermore, we do not assume any interpolation, and hence any mesh-like input data are accepted. Thanks to these nice properties we can effectively construct a COT-representation from discrete flow data. In the second part of our series presentations, we will provide quick explanations for the algorithm and demonstrate usage of a library “paclose” for TFDA.

1This work is supported by Japan Science and Technology.
11:49AM S12.00007 Topological Flow Data Analysis Part 3 - Foundation of describing flows on 3D spaces¹, TOMOO YOKOYAMA, Kyoto University of Education / Japan Science and Technology Agency, TAKASHI SAKAIJO, Kyoto University, TOMOKI UDA, Tohoku University — Topologies of flows on 3D spaces are quite complicated. Though every loop (e.g. periodic orbit) separates surfaces but chaotic stationary 2D flows on surfaces do not appear, no loops separate any 3D spaces but chaotic behaviors appear in stationary flows on 3D spaces. Moreover, periodic orbits are unknotted in 2D spaces but they can be easily linked in 3D spaces. Therefore we need the information both of chaos and of linking of infinitely many periodic orbits to analyze flows on 3D spaces. Hence it's very hard to describe topology of flows on 3D spaces in general. However, if there is uniformity (resp. symmetry) of flows, we can use slices of flows on 3D spaces to analyze the topology of such flows as medical doctors use X-ray photographs or computerized tomography images to analyze blood currents. On the other hand, the resulting flows on the slices are not incompressible even if the original 3D flow is incompressible. Thus we need to describe generic 2D flows. Therefore we constructed a classification of generic surface flows, which are called flows of finite type. In the third part of this sequential talks, we will explain the background of representations of 2D flows and introduce a representation of 2D flows of finite type and analyze topologies of flows on 3D spaces.

¹This work is supported by Japan Science and Technology.

12:02PM S12.00008 Topological equivalence between 3D buoyancy-driven and lid-driven cavity flows¹, SEBASTIAN CONTRERAS, Eindhoven University of Technology, IMAN ATAIEI, Delft University of Technology, MICHEL SPEETJENS, Eindhoven University of Technology, CHRIS KLEIJN, MARK TUMMERS, Delft University of Technology, HERMAN CLERCX, Eindhoven University of Technology — The present study concerns Lagrangian transport and (chaotic) advection in three-dimensional (3D) flows in cavities under steady and laminar conditions. The main goal is to investigate topological equivalences in 3D streamline patterns and their response to nonlinear effects between flow classes driven by different forcing. To this end we consider two classical systems that are important in both natural and industrial applications: a buoyancy-driven flow (laterally-heated configuration) and a lid-driven flow governed by the Grashof (Gr) and the Reynolds (Re) numbers, respectively. Symmetries imply fundamental similarities between the streamline patterns of these flows. Moreover, nonlinearities induced by buoyancy (increasing Gr) in the buoyancy-driven flow versus fluid inertia (increasing Re) and forcing protocol in the lid-driven flow cause similar bifurcations of the flow topology. These analogies imply that Lagrangian transport is governed by universal mechanisms and differences are restricted to the manner in which these phenomena are triggered. Experimental validation of key aspects of the Lagrangian dynamics is carried out by particle image velocimetry and 3D particle-tracking velocimetry.

¹S.C. acknowledges financial support from CONACYT Mexico.

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S13 Convection and Buoyancy-driven Flows: Free Convection 304 - Olga Shishkina, Max-Planck Institut

10:31AM S13.00001 Multiscaling analysis of buoyancy-driven turbulence in a differentially heated vertical channel, TIE WEI, New Mexico Tech — A multiscaling analysis is presented for the turbulent flow and heat transfer in a differentially heated vertical channel (DHVC). Based on the characteristics of force balance, a three-layer structure is proposed for the mean momentum balance (MMB) equation. In Layer I, a viscous inner layer adjacent to the wall, the force balance is between the viscous force and the buoyancy force. In Layer II, the outer layer, the force balance is between the Reynolds shear force and the buoyancy force. A multiscaling analysis of the MMB equation is developed for the inner and outer layers. A proper scale for the Reynolds shear stress is found to be \( \tau_u U_{\text{max}} \) where \( \tau_u \) is the friction velocity and \( U_{\text{max}} \) is the maximum streamwise velocity. The structure for the mean heat (MHB) equation can be divided into three layers based on the characteristics of the diffusional and turbulent heat flux. The outer scaling of the MHB equation in a DHVC is similar to passive scalar transport in forced convection. However, the inner scaling for the thermal inner layer in a DHVC is distinctly different from that in forced convection. The multiscaling analysis of the MMB and MHB equations agree well with the direct numerical simulation data of DHVC.

10:44AM S13.00002 Superstructures in turbulent thermal convection in slender cells¹, OLGA SHISHKINA, LUKAS ZWIRNER, Max Planck Institute for Dynamics and Self-Organization, Goettingen — The large scale circulation (LSC) in turbulent Rayleigh–Bénard convection and inclined convection of small-Prandtl-number fluids in slender geometries (cylindrical containers of the diameter-to-height aspect ratio smaller than one) is investigated. We investigate in detail the structures of the so-called single-roll and multiple-roll LSCs, their strength and path lengths and their relation to the strength of the volume-averaged heat transport in the system. The problems of the LSC extraction in experiments and numerical simulations and interpretation of the different LSC modes will be also discussed.

¹This work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grants Sh405/7 (SPP 1881 “Turbulent superstructures”) and Sh405/4 (Heisenberg fellowship). We acknowledge the Leibniz Supercomputing Centre (LRZ) for providing computing resources.

10:57AM S13.00003 Turbulent natural convection in a cavity with a free surface under non-Oberbeck-Boussinesq conditions, WILLIAM HAY, MILTIADIS PAPALEXANDRIS, Université catholique de Louvain — Turbulent natural convection of liquids is encountered in many environmental and industrial applications. For example, in the oceans, the buoyant flow of water due to temperature and concentration gradients has been studied extensively over the years. More recently, highly turbulent flows of a similar nature are being studied to better understand thermal mixing in nuclear spent fuel pools under accidental conditions. In such a case, the Rayleigh number can be as high as \( 10^{14} \). Moreover, due to significant temperature variations, the water transport properties can no longer be considered as constant. Therefore, the validity of the Oberbeck-Boussinesq approximation for the flows of interest becomes questionable. In this talk we present results from both direct numerical and large-eddy simulations of a cubic domain, periodic in the longitudinal direction, with a free-slip upper boundary and with water as the working fluid under non-Oberbeck-Boussinesq conditions. Over a series of simulations at increasing Rayleigh number we assess the impact of the free-slip boundary and variable fluid properties on the turbulent flow statistics.
11:10AM S13.00004 Influence of shroud–chimney configuration on heat transfer from horizontal cylinder: Experimental and numerical investigation. 1, MOHAMMED A. SAMHA, Rochester Institute of Technology - Dubai, OMAR ALI, Zakho University, MOHANAD TAH M. ALI, Rochester Institute of Technology - Dubai — In our prior study, the novel shroud–chimney configuration SCC (semicircular shrouds and expended chimney) has been numerically demonstrated to passively augment natural convection heat transfer from a horizontal cylinder. However, in order to implement such a configuration for practical utilizations, the heat flow properties must be experimentally observed and understood. In this work, well-controlled experiments are carried out to show the impact of SCC on the heat transfer from a horizontal cylinder subjected to constant measured heat fluxes. Circumferential temperature measurements at the cylinder surface, shrouds and ambient are performed using thermocouples. The emissivity of the cylinder is measured using a thermal camera that is needed for estimating the heat radiation. All presented cases are numerically simulated for validation. The measurements show that SCC promotes the convection heat transfer from the cylinder agreeing well with the numerical results. This validates the capability of this simple inexpensive passive method for practical uses. Furthermore, a parametric study is presented to show the optimum range of the design parameters for the best SCC performance.

1Dubai Silicon Oasis Authority (DSOA) Grants

2

11:23AM S13.00005 Turbulent thermal convection: the differential heating effects. 1, PHILIPP REITER, Max-Planck Institute for Dynamics and Self-Organization, Göttingen, RODION STEPANOV, Institute of Continuous Media Mechanics, Perm, OLGA SHISHKINA, Max-Planck Institute for Dynamics and Self-Organization, Göttingen — A significant class of geophysical and astrophysical flows are excited by temperature variations along a surface of the fluid layer. Using 3D direct numerical simulations, we study the effects of different temperature and velocity boundary conditions in Rayleigh-Benard convection and horizontal convection, under the requirement that the area-averaged temperature of the heated and cooled plates are kept constant. For these systems we analyze the global flow structures and the heat transport and investigate their dependences on the particular parameters of the boundary conditions. To explain our findings, we exploit a decomposition of the flow fields into the mean and fluctuation components. For some configurations (in horizontal convection) we also extract different Rayleigh-number regimes and present stability thresholds above which the flow exhibits characteristic global structures and give theoretical explanations for the underlying mechanisms of these structures.

1This work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grants Sh405/10 and Sh405/4 (Heisenberg fellowship) and SPP 1881 (“Turbulent superstructures”). We acknowledge the Leibniz Supercomputing Centre (LRZ) for providing computing resources.

11:36AM S13.00006 The Effect of Horizontal Buoyancy on Turbulent Thermal Convection. LU ZHANG, KE-QING XIA, Southern University of Science and Technology, The Chinese University of Hong Kong — We study the effect of horizontal buoyancy on heat transport in turbulent thermal convection system. Experimentally, a condition of increasing horizontal buoyancy (Ra_H = \alpha g \sin \beta \Delta T^3 / \nu \kappa) under fixed vertical thermal driving strength (Ra_v = \alpha g \cos \beta \Delta T^3 / \nu \kappa) is achieved by simultaneously tilting the convection cell by an angle \beta and increasing the imposed global temperature difference \Delta. Furthermore, we propose a vector formed Nusselt number Nu to quantify the global heat transport. For fixed vertical buoyancy, we find that the vertical heat transport increase monotonically with the horizontal buoyancy and the horizontal heat transport is also non-negligible. We also conduct direct numerical simulations, the results of which confirm our experimental findings and highlights the richness in convective transport.

1We gratefully acknowledge support of this work by the Research Grants Council of IKSAR (No. CUHK14301115 and 14302317), a SUSTech Startup Fund and the support by Center for Computational Science and Engineering of SUSTech.

11:49AM S13.00007 Numerical analysis of 3-D laminar natural convection heat transfer from solid vertical cylindrical heat sinks with straight longitudinal fins and comparison with horizontal heat sinks of the same configuration. DIBYENDU KONAR, Research Scholar, Dept. of Mechanical Eng., Indian Institute of Technology, Kharagpur - 721302, INDIA., SHUBHAM MISHRA, Research Scholar, Dept. of Mechanical Eng., Indian Inst of Technology, MOHAMMAD ASIF SULTAN, VIDYADHAR KARLAPALEM, Research Scholar, Dept. of Mechanical Eng., Indian Institute of Technology, Kharagpur - 721302, INDIA., SUBHRANU ROY, Professor, Dept. of Mechanical Eng., Indian Institute of Technology, Kharagpur - 721320, INDIA. — Of the various challenges facing the electronics industry, keeping components cool is utmost important, since overheating reduces the reliability and operating life of a device. Among various cooling methods, natural convection has been deemed fit owing to its distinctive advantages over forced convection. Longitudinal fins are generally used for vertical cylindrical heat sinks since the geometry eases air flow between successive fins. In the present study, CFD analysis of conjugate natural convection from such heat sinks have been carried out in the laminar regime. 15 different heat sinks modelling heat sinks of LED bulbs of dimensions obtained from literature have been developed. Effect of non dimensional fin spacing, fin length and Rayleigh number on the thermal resistance of the heat sink have been studied and values compared with those of horizontal sinks available in literature. It is observed that compared to the horizontal sink, the thermal resistance of the vertical sink is decreased by different percentages for different configurations. Contour plots to capture temperature profiles around the vertical heat sink have been developed. Correlations for Nusselt number have been formulated which are found to show appreciable agreement with computational data.

12:02PM S13.00008 Linear stability analysis of radiatively-driven convection in a lake. 1, TODD CHRISTOPHER, WILFRIED COENEN, STEFAN LLEWELLYN SMITH, University of California, San Diego — Observations of springtime warming of ice-free Lake Superior show that an instability arises each day at the surface and propagates down though the water column on a time scale of hours before restratification occurs at night. This situation is modeled using the Boussinesq approximation, leading to a Rayleigh-Benard-like configuration, except with a periodically-varying forcing term to capture the diurnal heat flux into the system. The forcing term is considered first as either a varying temperature or a varying heat flux at the boundary, but radiative heating in the bulk of the fluid is also investigated. A linear stability analysis of the system leads to a Floquet differential equation eigenvalue problem for the critical Rayleigh number. This problem is solved using a pseudospectral numerical method. The results found agree with the literature where applicable, while extending previous work to cover different boundary conditions and different forms for the forcing term.

1This work is partly supported by NSF award OCE-1829919.
10:31AM S14.00001 Fluid-Structure Interactions of Structure-Borne Traveling Waves, PATRICK MUSGRAVE, AUSTIN PHOENIX, U.S. Naval Research Laboratory — Structure-Borne Traveling Waves (SBTWs) are a promising means of underwater propulsion, but the coupled fluid-structural dynamics are poorly understood. SBTWs operate by taking advantage of a structure’s inherent modal properties (mode shapes and natural frequencies). This enables traveling wave generation using two actuation points instead of requiring a large array of actuators. However, the fundamental reliance of SBTWs on structural properties also introduces coupling with the surrounding fluid. This study extends SBTW generation to underwater and accounts for the two-way coupled, fluid-structure interactions. SBTWs are analytically and experimentally generated on a slender, cantilever beam in quiescent fluid. An analytic model is developed coupling a linear Euler-Bernoulli beam with a potential flow solution for the surrounding fluid. The structural response is solved via a Galerkin-type solution. SBTWs are then generated by applying multi-input forcing to the coupled system. The model is experimentally validated against a cantilever beam in quiescent water. Experimental SBTWs are generated using flush-mounted piezoelectric actuators and the structural response measured using non-contact scanning laser vibrometry. Analytic and experimental SBTWs are compared at several frequencies and with varying wave forms (i.e. wavelength and wave speed). The results demonstrate that SBTWs can be generated in water and the fluid-structure interactions accurately captured.

10:44AM S14.00002 Fluid-Structure Simulations of a Flexible Heaving Airfoil, JONATHAN CAPPOLA, DAVID MACPHEE, The University of Alabama — Motivated by experimental studies in biomimetic propulsion, a computational framework is used to simulate the heaving of a two-dimensional NACA airfoil with chordwise flexibility. A strongly-coupled fluid-structure interaction solver is developed using finite-strain solid deformation and a translating reference frame within the OpenFOAM framework. Using this model, we investigate any possible performance improvements of a flexible airfoil over the rigid in varying oscillating frequency and material elastic modulus. Flow structures and stress configurations are analyzed and discussed as related to simulated thrust and lift enhancements.

10:57AM S14.00003 Numerical and Experimental Investigation of Oscillating Flexible Foils with Application in Energy Harvesting, KIANA KAMKRANI FARD, PENGLEI MA, MICHAEL PRIER, JAMES LIBURDY, Oregon State University — The effect of relative leading edge motion of a flapping airfoil in energy harvesting regime is studied in reduced frequencies of \( k = f c / U = 0.06 \sim 0.14 \), pitching amplitude of \( \theta = 70^\circ \) and heaving amplitude of \( h_0/c = 0.5 \). A low order discrete vortex model with a vortex shedding criterion at the leading edge is used to estimate the transient lift force and the results are compared to 2-D CFD and direct force measurements via load cell. Positive leading edge motions in which the leading edge rotates in the same direction as the pitch angle and reduces effective angle of attack, are shown to improve the heaving power coefficient and efficiency compared to the rigid case by shifting the primary peak later in the stroke where the heaving velocity is higher. Whereas negative leading edge motions, in which the leading edge rotates in the opposite direction as the pitch angle and increases effective angle of attack, are shown to negatively influence the heaving power coefficient and efficiency compared to the rigid case, by reducing the primary peaks.

11:0AM S14.00004 An Alternative Geometry for a Galloping Energy Harvester, SAM TUCKER HARVEY, PETR DENISSENKO, IGOR KHOVANOV, University of Warwick — Interest in aeroelastic energy harvesters has grown substantially in recent years due to their potential for low maintenance and low cost energy solutions, particularly with regard to autonomous electrical devices, such as wireless sensors. The development of aeroelastic energy harvesters to date has focused mainly on the flutter of airfoils, the galloping of prismatic structures and vortex induced vibrations as a means to generate energy. In this work an alternative geometry for a galloping energy harvester, initially inspired by the trembling of aspen leaves in barely noticeable flows, is investigated in two alternative configurations. The dynamics of a prototype device have been characterised experimentally with the use of a motion tracking system, while the flow patterns generated around the device have been evaluated by smoke wire visualisation and particle image velocimetry (PIV). In the second configuration the presence of a leading edge vortex is found to coincide with higher potential energy harvesting performance. The interaction of multiple harvesters within the flow field is also demonstrated to result in phase locking synchronisation.

11:23AM S14.00005 Unsteady loads mitigation using flexible wings, GABRIELE PISSETTA, IGNAZIO MARIA VIOLA, School of Engineering, University of Edinburgh — In nature, fluid flows are inherently unsteady, and any wing-like device immersed in them experiences loads fluctuations. In some cases, these fluctuations may result in fatigue failures, and thus they strongly affect the reliability of the whole device. An effective control strategy would consist in a passive device capable of applying fast, local control action. This can be achieved using a flexible structure. In this presentation, we consider the loads on a tidal turbine operating in a shear flow, and we introduce a novel blade design to reduce the load fluctuations. We show that a blade with a flexible trailing edge can mitigate the fluctuations of the blade root bending moment, without affecting the mean torque, and thus the power generated by the turbine. Using a numerical method based on the semianalitical model, we model the loads flexibility as a torsional spring, and we perform a parametric study to identify the optimal spring parameters. The dynamic analysis of the system shows that the fluctuations of the root bending moment can be reduced by 93%. Our results prove the potential of a flexible structure to alleviate the loads fluctuations arising on a wing in an unsteady flow, and they underpin the development of more sophisticated models of flexible wings.

11:36AM S14.00006 A numerical study of a vertical axis turbine with chordwise-flexible blades operating at low tip speed ratios, PIERRE-OLIVIER DESCOTEAUX, MATHIEU OLIVIER, Universit Laval — This talk will present high-fidelity numerical simulations allowing the performance evaluation of a Vertical-Axis Turbine (VAT) equipped with chordwise-flexible blades. The simulations are carried out with a partitioned Fluid-Structure Interaction (FSI) code in which an in-house structural finite-element solver is linked to a finite-volume flow solver based on the OpenFOAM library. The idea behind this study is to take advantage of the strongly changing flow conditions acting on the blades when the VAT operates at low tip speed ratios. In such cases, the unsteadiness of flow forces can be used to alter the shape of the blade. This analysis will show under which conditions it is possible to increase the efficiency of a VAT by allowing passive foil deformations at a high Reynolds number. The FSI effects related to the flexibility and inertia of the blade will be investigated and the mechanisms that allow efficiency improvements, such as stall mitigation, will be described and compared against rigid-blade VAT operating in the same regime.

1This work was supported by Engineering and Physical Sciences Research Council grant number EP/N508796/1.

1This work is supported by EPSRC, grant number EP/L016680/1.
11:49AM S14.00007 Fully-resolved wave-structure interaction simulations of a two-dimensional submerged point absorber with three degrees of freedom, PANAGIOTIS DAFNAKIS, Politecnico di Torino, AMNEET PAL BHALLA, San Diego State University, GIULIANA MATTIAZZO, GIOVANNI BRACCO, Politecnico di Torino, SAN DIEGO STATE UNIVERSITY TEAM, POLITECNICO DI TORINO TEAM — A fully-resolved wave structure interaction (WSI) framework is developed to simulate a submerged point absorber. The WSI model is based on the fictitious domain Brinkman penalization method in which the solid body is treated as a porous body of vanishing permeability. For validating the model, forced damped-oscillation of a cylinder in various damping regimes along with several grid convergence studies are performed. The WSI model is compared against Cummins equation based Simulink model to demonstrate the differences between the potential flow theory and the nonlinear Navier-Stokes based methodologies. Time domain simulations are carried out for one, two, and three degrees of freedom buoy in order to analyze the surge, heave and pitching motion of the device using two methods. Furthermore, the WSI model is used to calculate the conversion efficiency of the point absorber for various wave and device parameters.

Tuesday, November 26, 2019 10:31AM - 12:02PM — Session S15: Compressibility Effects 310 - Simo A. Makiharju, UC Berkeley

10:31AM S15.00001 Experimental study on the effect of nozzle flexibility on the evolution of a starting liquid jet¹, DAEHYUN CHOI, HYUNGMIN PARK, Seoul National University — Jet flow is universal phenomena in propulsion, cooling, mixing, and coating. Compared to a jet flow issued from a rigid nozzle, the effect of deformable nozzle has not been understood well. The purpose of this study is to characterize how the flexibility of the nozzle affects the evolution of a starting jet. The nozzle shape is thin-walled circular pipe, and is made of silicone rubber. The flexibility of the nozzle is adjusted as stiff, flexible, and highly flexible cases. The piston-motor system generates a starting jet (Reynolds number of 3,000) that accelerates initially and then reaches a constant speed in a time duration of 0.2 seconds. We use 2D particle image velocimetry (PIV), and 3D digital image correlation (DIC) for measuring the velocity field of the water jet and the deformation of the nozzle, respectively. We find that the nozzle experiences a sudden expansion process as the jet evolves, and gain in fluidic impulse is obtained. When the jet reaches a constant velocity, the nozzle exhibits periodic or static movements according to the flexibility, affecting the performance of the jet mixing, which is quantified by the entrainment rate. A discussion of a dimensionless number that governs the jet-flexible nozzle interaction will be finally discussed.

¹Supported by grants (2017R1A4A1015523, 2017M2A8A4018482, KCG-01-2017-02) funded by Korea government and Korea Coast Guard.

10:44AM S15.00002 Super jets: the breakup of liquid jets in super- and normal-fluid helium-4, NATHAN SPEIRS, KENNETH LANGLEY, King Abdullah University of Science and Technology, PETER TABOREK, University of California Irvine, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — We experimentally examine the breakup of super and normal-fluid liquid helium-4 in an atmosphere of its own vapor as the temperature is varied from 1.2 K to the critical point. The jets evolve through five different breakup modes over the large range of Ohnesorge number, Reynolds number, and gas-liquid density ratio that we examine. As jets approach and pass through the critical point the liquid phase vanishes and we see a transition from liquid breakup to turbulent gaseous structures. This unique parameter space allows us to delve into the underlying physics of the various breakup modes and propose new quantitative transition criteria supported by dimensional analysis and a large three-dimensional data set.

10:57AM S15.00003 Structures Within Sub- and Supersonic Submerged Gas Jets, JASON T. PARKER, SIMO A. MAKIHARJU, University of California, Berkeley — Sub- and supersonic submerged gas jets (SGJs) are used in steel making, underwater propulsion, and wastewater treatment. Prior experiments on SGJ dynamics have used optical techniques to study the gas-liquid interface, but to the authors’ knowledge no experimental data exists for the internal phase fraction distribution. Observations by previous investigators suggest that a turbulence peak may explain jet pinch-off (JPO) onset and location. It is unclear whether coherent structures within the SGJ appear and grow before JPO. Grow and Champagne (1971) discovered coherent structures on the surface of a gas jet in air, suggesting that coherent structures may also exist inside a SGJ. Furthermore, validation of numerical models has relied on comparisons between CFD models and high-speed footage that only shows the boundary of the SGJ. In addition to providing a measurement of the internal phase fractions to probe SGJ physics, the present experiments provide validation data for CFD models. The tested gas exit velocities span Mach 0.6 to 1.4. We utilize 13kHz X-ray densitometry located 10-15 nozzle diameters downstream of the orifice to coincide with the predicted turbulence peak and the location of most frequent JPO.

11:10AM S15.00004 Thrust and Flow Field Characteristics of Asymmetric Turbulent Pulsed Jets, CESAR LEOS¹, Member, ROBERT FREEMAN², None, ISAAC CHOUTAPALLI³, Member — An experimental study was conducted to study the effect of nozzle exit Mach number and the nozzle exit geometry on the characteristics of a free pulsed jet and pulsed jet ejector. Four converging nozzles of various exit geometry (circular, diamond, elliptic, and rectangular) were utilized to perform the study. The diameter of the axisymmetric circular nozzle is 50.8 mm and the asymmetric nozzles had the same exit area as that of the circular nozzle with aspect ratio 2:1. Both the thrust and flow field measurements were conducted for nozzle exit Mach numbers 0.10, 0.20 and 0.30 over a range of pulsing frequencies from 24Hz to 180Hz. The force measurements showed that, for a given nozzle geometry, the free pulsed jet and pulsed ejector thrust augmentor exhibited a weak dependence on the nozzle exit Mach number and a strong dependence on the pulsing frequency. Furthermore, strong evidence of thrust augmentation dependence on nozzle geometry, exit Mach number, and pulsing frequency was observed for the nozzle cohort. Flow field measurements using PIV showed that the phase-averaged and global flow characteristics of the free pulsed jet and pulsed ejector are dependent on pulsing frequency and nozzle exit geometry, with a weak dependence on the nozzle exit Mach number.

¹Graduate Research Assistant, Member APS
²Professor
³Associate Professor, Member APS
11:23AM S15.00005 Hybrid Large Eddy Simulation and Lagrangian Particle Simulation of Passive Scalar Mixing in a Supersonic Jet

YOUNG TAI, TOMOAKI WATANABE, KOJI NAGATA, Department of Aerospace Engineering, Nagoya University — Large eddy simulation combined with Lagrangian particle simulation (LES-LPS) is developed for predicting passive scalar mixing in supersonic turbulent flows. LPS solves a governing equation of passive scalar with notional particles, where the velocity of fluid particles is provided by LES while a molecular diffusion term is modeled by a mixing model. We propose a mixing volume model that computes the molecular diffusion term based on spatial averaging. In this model, the coarse-grained scalar dissipation rate is computed from a coarse-grained scalar gradient estimated from particles with an aid of a subgrid-scale model of the scalar dissipation rate. The LES-LPS with the proposed mixing model is applied to a supersonic planar jet with passive scalar transfer. The LES-LPS is evaluated by comparing the results with direct numerical simulation (DNS) databases. In the present study, the number of mixing particles \( N_{m} \) is between 8 and 24. A mean scalar profile is well predicted by the LES-LPS for all of these parameters. However, root-mean-squared scalar fluctuations tend to increase with \( N_{m} \). Comparisons of other statistics, such as probability density functions, confirm that the LES-LPS with \( N_{m} = 12 \) well predicts the passive scalar in the supersonic turbulent jet.

1Supported by the high-performance computing system in Nagoya University

11:36AM S15.00006 Investigating the Turbulence Physics of a Supercritical Carbon Dioxide Round Jet

JULIA REAM, Florida State University, MARC HENRY DE FRAHAN, MICHAEL MARTIN, SHASHANK YELAPANTULA, RAY GROUT, National Renewable Energy Laboratory — In this investigation, we study Supercritical Carbon Dioxide (sCO2) jets to gain a better understanding of the underlying physics associated with supercritical fluid flow. sCO2 is a promising working fluid for advanced cycles including those for power generation (e.g., Brayton cycle) due to increased power density. Open questions remain about how the fundamental physics of these flows are impacted by non-ideal variation in the physical properties. We use a second order finite volume method with adaptive mesh refinement as implemented in the first-principles simulation code PeleC to establish the impact of a cubic equation of state on turbulent flow physics. The Soave-Redlich-Kwong equation of state is used to close the system of equations. We simulate a sCO2 turbulent round jet at 600 K and 10 MPa. We then examine velocity and Reynolds stress profiles at different downstream locations and contrast these with established theory. These conditions are above the critical point of 304.25 K and 7.39 MPa, where new insight is needed for engineering design. We then explore cases in which the temperature of the jet and that of the ambient fluid differ, capturing effects of widely varying thermal properties of supercritical fluids.

1Supported by National Science Foundation Mathematical Sciences Graduate Internship Program and the Exascale Computing Project

11:49AM S15.00007 Effect of Mach Number and Stagnation Temperature on the Performance of a Pulsed Jet Ejector

GREG ACOSTA, ROBERT FREEMAN, ISAAC CHOUTAPALLI, The University of Texas - Rio Grande Valley — An experimental study was carried out to investigate the effect of Mach number and stagnation temperature on the performance characteristics of a pulsed jet ejector. The nozzle exit Mach number was varied from 0.3 to 0.8, and the temperature ratio \( \Gamma \) was varied from 1.04 to 1.77. The thrust measurements on a free pulsed jet showed that the thrust varied marginally with the primary jet Mach number at a given pulsed frequency. The thrust measurements on the pulsed jet ejected showed that the thrust augmentation ratio is a weak function of the primary jet Mach number and is mainly dependent on the ejector area ratio. An increase in the temperature ratio for the pulsed jet ejector with the primary jet operating at Mach 0.80, resulted in a decrease in thrust augmentation ratio. The global flow field of the free pulsed jet showed that within the subsonic regime, the primary jet Mach number has a minimal effect on the centerline velocity decay and jet spreading for a given Strouhal number. The phase averaged flow field showed that the vortex ring circulation varies minimally with the jet Mach number for a given Strouhal number. The turbulence field showed that the magnitude of the peak turbulent stresses decreased as the jet Mach number is increased.

Tuesday, November 26, 2019 10:31AM - 12:15PM Session S16 Tidal Energy

4c3 - Arindam Banerjee, Lehigh University

10:31AM S16.00001 A hybrid approach for power prediction of tidal stream turbine using transient blade element momentum theory

CONG HAN, ASHWIN VINOD, ARINDAM BANERJEE, Lehigh University, THOMAS LAKE, MICHAEL TOGNERI, IAN MASTERS, Swansea University — Blade Element Momentum (BEM) method is traditionally used to evaluate hydrodynamic performance of wind/tidal turbine blades. Steady-state BEM predictions are based on the assumption that the inflow is uniform at the rotor plane. Majority of tidal energy sites have high levels of Free-Stream Turbulence (FST) and power prediction in those scenarios needs to account for fluctuations in the free-stream. In addition, any variation in hydrofoil (lift-drag) dynamics due to FST needs to be also accounted for. A hybrid approach was implemented to predict the performance of a tidal turbine model in a water tunnel test section. Lift/drag characteristics of a SG6043 hydrofoil was measured in a water tunnel facility fitted with an active grid turbulence generator at Lehigh University. The inflow properties measured using an ADV along with lift/drag measurements are used as an input to the BEM code. Validation of simulation and verification of the hybrid model is done by comparing the transient BEM predictions to the experimental torque-thrust measurements with a scaled turbine model under elevated FST conditions.

1The authors acknowledge financial support from NSF Fluid Dynamics (Grant 1706358)

10:44AM S16.00002 Wake dynamics behind two closely spaced vertical axis turbines

CATHERINE WILSON, VALENTINE MUHAWÉNIMANA, STEPHANIE MUELLER, PABLO OÙRO, Cardiff University, ALDO BENAVIDES, CARLOS DUQUE, Universidad Nacional de Colombia, CARDIFF UNIVERSITY COLLABORATION, UNIVERSITY OF WARWICK COLLABORATION — The technology for harnessing kinetic energy from rivers via turbines has evolved at a slower pace than in wind or tidal environments. Widely adopted in these fields due to the high energy conversion rates, vertical axis turbines operate at high rotational speeds in high velocity environments, which can drastically impact ecosystems. Alternatively, vertical axis turbines are designed to operate at lower rotational speeds, and with the advantage of their rectangular cross-section, efficiently extract kinetic energy from river streams while reducing environmental impact. This research tested two small-scale vertical axis turbines in a hydraulic flume, measuring their wakes up to 10 diameters downstream and across the flume width. The three-bladed devices rotated at a constant speed equivalent to their optimum energy conversion rate. Flow velocity measurements results in terms of mean velocities and turbulence fluctuations show that the individual wakes merge into a single low-velocity wake and this directly affects the flow processes involved. Flow measurements also captured the tip vortices developed during the upstroke and downstroke rotation of the turbines. Comparisons of these wake dynamics were analysed for two lateral spacings of 1.5 and 2.0 turbine diameters.
turbine-wake interaction
interactions and hypothesize how operating conditions affect array performance. By contrasting configuration and control cases, we explore the hydrodynamic formulation of the model has a strong influence in both instantaneous and mean flow field, especially in the topology and dynamics of turbulent interactions between the flow and the turbine blades. In this study we present a comprehensive analysis of the hydrodynamic behaviour of the wake generated downstream a tidal turbine using three different actuator approaches to represent turbines in the flow: Actuator Disks Model (ADM), a model based on Blade-Element Momentum (BEM) theory and an Actuator Lines Model (ALM). Each computational turbine models interactions such as shear, large temporal fluctuations and turbulence. Through URANS tandem cross-flow turbine simulations, this study provides some physical insight into turbine-wake interactions and on the effect of flow perturbations on effective performance coefficients. Although the different types of perturbation affect significantly and differently the dimensional power of turbines, they are found to yield similar predictions for confined turbines. However, because of turbine-wake interaction, the characteristics of the local flow also include different types of perturbation such as shear, large temporal fluctuations and turbulence. Through URANS large tandem cross-flow turbine simulations, this study provides some physical insight into turbine-wake interactions and on the effect of flow perturbations on effective performance coefficients. Although the different types of perturbation affect significantly and differently the dimensional power of turbines, they are found to yield similar effective performance coefficients, which is quite encouraging for the future developments of simplified turbine models.

This research was partially supported by the supercomputing infrastructure of the NLHPC (ECM-02)

Numerical Simulation of a Drag-driven Vertical Axis Hydrokinetic Turbine in Open Channel Flow, JINJIN GAO, Hohai University / University of Minnesota, Twin cities, YUAN ZHENG, Hohai University, MICHELE GUALA, LIAN SHEN, University of Minnesota — A drag-driven vertical axis hydrokinetic turbine, partially embedded in a relatively shallow channel streambank, is expected to partially absorb the kinetic energy of the river. To study its performance and wake characteristics, numerical simulation of the turbine in an open channel flow is conducted. Large-eddy simulation of the flow in an arbitrarily complex domain involving moving or stationary boundaries is carried out to investigate the structure of turbulence in the wake of the turbine and optimize its performance. The complex turbine geometry, including the rotor and the cavity along the bank, is captured by the immersed boundary method. Coupled level-set and volume-of-fluid method is used to capture the deformable free surface. The power coefficients of the turbine at different angular velocities, and tip speed ratios, are calculated and compared against the experimental data. The simulation results reveal the wake flow structures generated by the turbine and will be used to improve the blade design.

Numerical Simulation of a Drag-driven Vertical Axis Hydrokinetic Turbine in Open Channel Flow, JINJIN GAO, Hohai University / University of Minnesota, Twin cities, YUAN ZHENG, Hohai University, MICHELE GUALA, LIAN SHEN, University of Minnesota — A drag-driven vertical axis hydrokinetic turbine, partially embedded in a relatively shallow channel streambank, is expected to partially absorb the kinetic energy of the river. To study its performance and wake characteristics, numerical simulation of the turbine in an open channel flow is conducted. Large-eddy simulation of the flow in an arbitrarily complex domain involving moving or stationary boundaries is carried out to investigate the structure of turbulence in the wake of the turbine and optimize its performance. The complex turbine geometry, including the rotor and the cavity along the bank, is captured by the immersed boundary method. Coupled level-set and volume-of-fluid method is used to capture the deformable free surface. The power coefficients of the turbine at different angular velocities, and tip speed ratios, are calculated and compared against the experimental data. The simulation results reveal the wake flow structures generated by the turbine and will be used to improve the blade design.

Cross-Flow Turbine Array Interactions, ISABEL SCHERL, BENJAMIN STROM, STEVEN L. BRUNTON, BRIAN L. POLAGYE, University of Washington — Cross-flow turbines, also known as vertical-axis turbines, use blades that rotate about an axis perpendicular to the incoming flow to convert the kinetic energy in moving fluid to mechanical energy. Arrays of cross-flow turbines with optimized geometries and control strategies have been shown to out-perform geometrically-identical turbines in isolation. In this work, the performance of a two-turbine array in a recirculating water channel was experimentally optimized across sixty-four unique array configurations. For each configuration, turbine performance was optimized using tip-speed ratio control where rotation rate for each turbine is optimized individually and using coordinated control where we operated the turbines at equal rotation rates and optimized the rotation rate and phase difference between the two rotors. By contrasting configuration and control cases, we explore the hydrodynamic interactions and hypothesize how operating conditions affect array performance.

Reliability of effective performance coefficients in the context of turbine-wake interaction, OLIVIER GAUVIN-TREMBLAY, GUY DUMAS, CFD Laboratory LMFN, Universit Laval — In the planning of turbine array deployment and for the performance prediction of its constituent turbines, the use of effective performance coefficients within the simplified turbine models is found to be very useful. Instead of being based on the far field upstream velocity as conventional drag and power coefficients, the effective coefficients are based on a local velocity, representative of the local conditions experienced by each turbine in the array. It has already been shown that effective performance coefficients take inherently into account blockage effects and allow good performance predictions for confined turbines. However, because of turbine-wake interaction, the characteristics of the local flow also include different types of perturbation such as shear, large temporal fluctuations and turbulence. Through URANS tandem cross-flow turbine simulations, this study provides some physical insight into turbine-wake interactions and on the effect of flow perturbations on effective performance coefficients. Although the different types of perturbation affect significantly and differently the dimensional power of turbines, they are found to yield similar effective performance coefficients, which is quite encouraging for the future developments of simplified turbine models.

The authors acknowledge the Natural Sciences and Engineering Research Council of Canada (NSERC) for their financial support as well as Compute Canada for their supercomputer allocation.
10:02PM S16.00008 Performance Assessment of a Wells Turbine with Morphing Blades, KELLIS KINCAID, DAVID MACPHEE, The University of Alabama — Wells turbines are often used to harvest energy from ocean waves when paired with an oscillating water column. Due to the necessity of self-rectifying characteristics in this design, blades are typically mounted normal to the flow direction, resulting in a narrow effective operating region and unfavorable attack angles as flow rate through the turbine increases. Various methods have been proposed to solve this issue, including guide vanes, active and passively actuated blades, and a static blade setting angle which takes advantage of the asymmetric nature of the reversing flow through the turbine in realistic operating conditions. In this work, we model a flexible blade with a static blade setting angle which takes advantage of the asymmetric nature of the reversing flow through the turbine in realistic operating conditions. The model uses a solver based in the OpenFOAM framework to investigate any performance gains realized by incorporating a flexible or “morphing” trailing edge to the turbine blades. With this modification, blades can deflect passively due to aerodynamic forces, resulting in lower effective angles of attack, higher torque output, and delayed onset of stall. In this work, simulation results for both flexible and rigid turbines are analyzed and compared, with discussions on flow structures and material deformations as they relate to turbine performance.

Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S17 Tandem and Flapping Wings 4c4 - Bo Cheng, Pennsylvania State University

10:31AM S17.00001 Effect of the rear wing size on the thrust performance of the two-dimensional tandem flapping wing,1 — SUNIL MANOHAR DASH, NISHANTH S, JIT SINHA, Indian Institute of Technology, Kharagpur, KIM BOON LUA, National Chiao Tung University, Taiwan, IIT KARHAGPUR - NCTU TAIWAN COLLABORATION — In this numerical study, the effect of the size of the rear wing on the thrust performance of 2D elliptical tandem flapping wing is investigated. Here, size ratio (SR), defined as the ratio of the chord of the rear to the front wing, is varied from 0.5 to 1.5 at an interval of 0.25 while keeping the aspect ratio (ratio of wing chord to thickness) AR—8, wing spacing (distance from trailing-edge of front wing to leading-edge of rear wing) λ— chord of the front wing, and phase angle ϕ = 0 deg constant. Reynolds number based on the chord and Strouhal number based on the excursion distance of the front wing are set as 5000 and 0.32, respectively. For different SRs, we notice that time-average (Ct) and peak transient (Ctp) thrust coefficients of the rear flapping wing in tandem configuration can be up to 80% higher compare to those of single flapping wing of same size. This enhanced thrust performance in tandem wing flapping is attributed to the constructive interaction of the shed vortices from the front wing with the leading-edge vortex of the rear wing. Note that the formation and interaction of the vortices is modified with SR. When SR increases, Ctp shifts towards starting of the flapping stroke and maximum Ctp and Ct are seen at SR=0.5 and SR=0.75, respectively.

1This work is supported by the Science and Engineering Research Board, India

10:44AM S17.00002 Multi-fidelity Kinematic Parameter Optimization of Flapping Airfoil, HONGYU ZHENG, FANGFANG XIE, YAO ZHENG, Zhejiang University, CENTER FOR ENGINEERING AND SCIENTIFIC COMPUTATION TEAM — We have constructed a multi-fidelity framework for kinetic optimization of flapping foil with inline motion. We employ multi-fidelity Gaussian process regression and Bayesian optimization to effectively synthesize the aerodynamic performance of flapping foil with the kinetic parameters under multi-resolution direct numerical simulations. The objective of this work is to demonstrate that the multi-fidelity framework can be used efficiently to discover optimal kinetic parameters of foil with desired aerodynamic performance using a limited number of expensive high-fidelity simulations combined with a larger number of inexpensive low-fidelity simulations. We efficiently identify the optimal kinetic parameters of asymmetric flapping foil with target aerodynamic forces in the design space of heaving amplitude, pitching amplitude, angle of attack (AOA) and the stroke angle. Specially, it is found that the AOA can affect the magnitude of the aerodynamic forces by violating the generation of leading-edge Vortex while its combination effect with the stoke angle can determine the attitude and trajectory of flapping airfoil.

10:57AM S17.00003 Physics-informed Predictive Model of Flapping Flight Aerodynamics using Gaussian Process Regression1 — YAGIZ BAYILG, KEEGAN HARRIS, The Pennsylvania State University, YU PAN, HAIBO DONG, University of Virginia, BO CHENG, The Pennsylvania State University — An accurate and computationally efficient model for predicting the aerodynamic force, moment and power of flapping flight can significantly advance the understanding animal flight and the design of bioinspired micro aerial vehicles. In this work, we develop such a predictive model based on Gaussian Process (GP), informed by quasi-steady aerodynamic model and trained by Computational Fluid Dynamics (CFD) simulation data for a wide range of flapping wing kinematics. The GP receives instantaneous wing kinematics as the input and uses statistical inference methods to predict the resulting forces, moment and power. The training set consists of a nominal wing trajectory and a set of trajectories highlighting a deviation from the vanilla trajectory in one particular kinematic feature. The resulting GP model is tested on a separate set of wing trajectories with a mixed change of kinematic features and is shown to provide more accurate predictions than the conventional quasi-steady models. The accuracy of the predictions relies on the proximity to the training set, and for a relatively wide range of trajectories, they show excellent agreement with the CFD results. The GP model also provides uncertainty information, indicating the regions where the prediction has a high variance.

1 NSF CMMI 1554429; ONR MURI N00014-14-1-0533

11:10AM S17.00004 Aerodynamic stability analysis of the dual-wing flapping flight for the large birds, TENGYU HOU, School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai, 200240, China, YANG XIANG, HONG LIU, J. C. Wu Center for Aerodynamics, School of Aeronautics and Astronautics, Shanghai Jiao Tong University, Shanghai, 200240, China — By dual-wing flapping, large birds obtain lift and thrust to sustain flight. Insects obtain good maneuverability and dynamic stability through high frequency flap. The airliners show good performance based on the principle of steady aerodynamics. Different from the two examples before, birds have a lower flapping frequency to maintain their stability for forward flight. Through the observation of bird flap behavior and human speculation, we can easily generate the idea that the bird’s flap can show stability only within a certain range of flapping frequency. In this article, we model birds based on a quasi-steady mathematical model, and observe the stability of the mathematical model by applying different perturbation excitations to the model. The results of theoretical analysis show that for a given parameter model, it is possible to maintain stable flight only within a certain range of flapping frequency. Moreover, we find that the model can resist disturbances of different degree in different frequency ranges. Then, we compare the dual-wing to the single-wing model. Our work shows that the flight stability of birds and the flapping frequency are closely related, which may provide effective theoretical guidance for the design of flapping wing air vehicles.
Rolling and twisting of finite foil

11:23AM S17.00005  ANDHINI NOVRTA ZURMAN NASUTION, BHARATHRAM GANAPATHISUBRAMANI, GABRIEL D. WEYMOUTH, University of Southampton — Strip theory in flapping foils, i.e. reconstructing a 3-dimensional (3D) behavior from a spanwise sequence of 2D strips, has a potential to give fast predictions but the modeling errors are poorly understood. In this work, finite foils with an elliptic tip are simulated with a sinusoidal motion of pure rolling and combined rolling-twisting. Their spanwise cross-sections are compared with 2D simulations of the same kinematics. The 3D pressure distributions are strongly correlated with the 2D, but have very different amplitudes. The spanwise velocity of the cross-sections also increases with the local velocity caused by the kinematics. The pure rolling motion indicates that maximum lift coefficient ($C_L_{max}$) at each cross-section augments linearly with local velocity square except at the tip. It is discovered that the slope of $C_L$ along the span declines as the aspect ratio is reduced with the same scaling as Prandtl’s finite-wing theory. Similar behavior is found by Green and Smits [2008] to scale the thrust coefficients of pitching panel for different aspect ratios. With these discoveries, the 2D $C_L_{max}$s can be corrected with the scaling to find their corresponding 3D cross-sections at different aspect ratios, enabling quantitative strip theory predictions of 3D flapping flight.

Vortex models for the unsteady aerodynamics of tandem foils

11:36AM S17.00006  JAVIER ALAMINOS-QUESADA, University of Malaga, JEFF ELDREDGE, University of California, Los Angeles — Configurations of multiple wings and fins arise in various contexts, including biological flying and pitching, biologically-inspired and rotary wing vehicles, and formations of vehicles. Here, we present two-dimensional potential flow models for the separated flows of multiple unsteady foils. In the special case of a longitudinal tandem configuration, these models are evaluated and compared directly with high-fidelity numerical simulations at low Reynolds number. In addition, we have obtained the optimal configuration for an accelerating pair of foils with respect to the separation distance between them. Finally, we consider potential flow models of the trailing airfoil in which the effect of the leading airfoil’s wake is replaced by a time-varying vortexicity flux into the separated flow of the trailer.

A numerical study of flapping wings in tandem configuration at low Reynolds number

11:49AM S17.00007  GONZALO ARRANZ, OSCAR FLORES, MANUEL GARCIA-VILLALBA, Universidad Carlos III de Madrid — Direct numerical simulations of wings in - in line tandem configuration are presented. % The wings undergo a two-dimensional optimal kinematics. % This optimal motion is a combination of heaving and pitching of the airfoils in a uniform free-stream at a Reynolds number 1000 and Strouhal number 0.7. % The objective of the study is to analyze how three-dimensional effects influence the aerodynamic performance of the wings. % To that end, wings of two different aspect ratios, 2 and 4, undergoing the two-dimensional kinematics are considered. % Simulations show that the interaction between the vortical structures of the wings is similar to the 2D case. % However, it is found that 3D effects are detrimental in terms of hind-wing’s thrust generation. % On the contrary, the propulsive efficiency remains constant both in 2D and 3D, for both aspect ratios. % Simulations of flapping motion are also presented and compared to the previous wings in heaving motion. % It is found that aerodynamic forces and propulsive efficiency decrease when the wings are in flapping motion due to a sub-optimal motion of the inboard region of the flapping wings.

A numerical study of flapping wings in tandem configuration at low Reynolds number

12:02PM S17.00008  JOHANNES KISSING, BASTIAN STUMPF, Technische Universität Darmstadt, Fluid Mechanics and Aerodynamics (SLA), JOCHEN KRIEGSEIS, Karlsruhe Institute of Technology (KIT), Institute of Fluid Mechanics (ISTM), CAMERON TROPEA, Technische Universität Darmstadt, Fluid Mechanics and Aerodynamics (SLA) — In order to achieve higher manoeuvrability, novel micro-air vehicle designs adapt flapping wings from biological flight to realize hovering and forward flight. The increased lift of flapping wings is mainly due to the circulatory lift of the leading-edge vortex (LEV), which forms on the wing and induces transient lift as it grows. Subsequently, the unsteady lift drops when the vortex detaches from the airfoil. In order to attain higher overall lift, the current project aims to delay the detachment of the LEV by manipulating the flow field at topologically critical locations with a dielectric barrier discharge plasma actuator (DBD-PA). Time-resolved particle-image velocimetry measurements are used to characterize the flow field around a flat plate, which executes a combined pitching and plunging motion at a Reynolds number of 24,000, a reduced frequency of 0.48 and a Strouhal number of 0.1. A prolongation of the growth phase of the LEV, marked by a longer circulation-accumulation phase and a decreased convection of the LEV, is achieved by compressing secondary structures ahead of the main vortex. Temporal variations of the excitation onset are found to be the key factor between spatial control authority of the DBD-PA and the determined effectiveness of flow control.

Can we achieve statistically stationary Homogeneous Shear Turbulence?

10:31AM S18.00001  CHANDRU DHANAPANI, GUILLAUME BLANQUART, California Institute of Technology — Homogeneous shear turbulence (HST) is an idealized version of the shear turbulence observed in practical free shear flows, and can be simulated using simple computational domains. One of the numerically-efficient configurations to simulate turbulent flows is to use triply periodic domains. However, owing to the mean stream-wise velocity being non-homogeneous, periodic boundary conditions cannot be used along one of the directions. Several studies included shear periodic boundary conditions in the cross-stream direction. However, in these simulations, the turbulence statistics grew exponentially with time, whereas the turbulence observed in free shear flows is statistically stationary. The authors fixed this problem earlier by performing HST simulations with only shear production and neglected shear convection, thereby obtaining statistically stationary shear turbulence. The current study improves upon the previous simulations by including shear convection, by introducing an inflow/outflow in the cross-stream direction. The turbulence statistics reach a statistically stationary state, and the Reynolds shear stress and the anisotropy values agree very well with the results from experiments and simulations of mixing layers, planar jets, and round jets.

Tuesday, November 26, 2019 10:31AM - 12:02PM

Session S18 Turbulent General 400 - Robert Breidenthal, University of Washington
Benard convection boundary layer. ROBERT BREIDENTHAL, University of Washington — Density effects are known to be weak in the free shear layer, yet they are commonly assumed to be strong in the compressible boundary layer. This apparent inconsistency is addressed in a single model by assuming that the acoustic signaling speed rather than density variations always controls the physics, in both free shear and wall flows. The model assumes that turbulent transport by an eddy requires information to propagate across the diameter of that eddy during the period of one eddy rotation, the time interval of importance in the physics. The turbulent fluxes are dominated by these 'sonic' eddies, whose rotational Mach number is underdamped, and the density gradients are neglected. To investigate this effect, large eddy simulations (LES) are performed to propagate a two dimensional fire front using the level-set method which is advanced in time by a third-order Runge-Kutta algorithm. The heat flux generated from the fire is fed back to LES as boundary condition and the effect of buoyancy on wind propagation is included to model a realistic fire weather interaction. From the simulation data, the effects of small scale turbulence structures and change in local atmospheric conditions on fire speed are analyzed. The LES simulation results are compared with the weather research and forecast (WRF) model simulation for wildland fire.

This work was supported by National Science Foundation, grant number 1836505

11:49AM S18.00007 On the Inertial Range Scaling in the High-R∞ Limit . CHRISTIAN KUECHLER, Max Planck Institute for Dynamics and Self-Organization, Goettingen, Germany, GREGORY P. BEWLEY, Cornell University, Ithaca, USA, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization, Goettingen, Germany — We investigate in a decaying laboratory flow the universal scaling laws Kolmogorov predicted in 1941 to emerge in the limit of infinite $R_\lambda$. In the past it has been found that this limit requires extreme $R_\lambda$, which are difficult to create in a well-controlled turbulent flow. The Variable Density Turbulence Tunnel (Bodenschatz et al., 2014) is the first wind tunnel capable of producing $R_\lambda > 5000$ and fully resolved inertial scales. It combines the low kinematic viscosity of pressurized SF6 and a unique mosaic-like active grid with individually controllable tiles (Griffin et al., 2019). To resolve the smallest scales present in the flow, we use Nanoscale Thermal Anemometry Probes developed and generously provided by Princeton University (e.g. Bailey et al. (2009), Vallikivi et al. (2014)). We present results that are consistent with Kolmogorov scaling and structure functions differ from conventional scaling laws of isotropic turbulence when studying the logarithmic derivatives of these statistics. However, these local scaling exponents approach a universal form when $R_\lambda > 2000$. We show that these results are well-described by the generalized self-similar spectrum of decaying turbulence introduced by Yang et al. (2018).


10:31AM S19.00001 A unified AMR framework for multiphase flow and fluid-structure interaction problems with both non-subcycling and subcycling . YADONG ZENG, LIAN SHEN, University of Minnesota — In the present work, we developed a unified structured adaptive mesh refinement (SAMR) framework for multiphase flow and fluid-structure interaction problems. The coarsest grid covers the whole computational domain, and is dynamically refined onto higher levels to obtain higher resolution in local regions. Both non-subcycling, where a uniform time step is employed for all variables on composite levels, and subcycling, in which variables on different levels advance in time at different rates, are being refined in the SAMR framework. While the former is easier to implement, the latter can significantly reduce the computational time while maintaining the same order of accuracy. For multiphase flow problems, the level set function is used to capture the interface, and re-initialization across different levels is applied. For fluid-structure interaction problems, a direct forcing immersed boundary method is coupled with SAMR. Validation cases are presented to demonstrate the accuracy and robustness of the proposed computational framework.
10:44AM S19.00002 An optimal sparse sensing approach for adaptive mesh refinement in unsteady flows. DANIEL FOTI, University of Memphis, SVEN GIORNO, KARTHIK DURAISAMY. University of Michigan — Complex physical flows are often characterized by coherent structures, which have a crucial role in turbulent flows. In order to accurately simulate the flow field, the coherent structures need to be adequately resolved with a sufficiently fine mesh. Furthermore, local mesh refinement in areas of interest can be employed to reduce time while preserving accuracy. While mesh adaptation techniques are well-established for steady flows, refinement methodology for unsteady, spatially-evolving flows is less straightforward. Residual and error-minimization based methods require precise definitions for spatio-temporal error, and feature or gradient based methods rely overly on user intuition of the flow, while adjoint-based methods can become expensive for finite volume methods. We introduce a novel approach for adaptive mesh refinement where selection is obtained similar to a computationally expedient discrete empirical interpolation method using rank-revealing QR. This method seeks optimal locations for grid adaptation from the basis of a proper orthogonal decomposition, which organizes velocity flow field features into optimal orthogonal modes based on energy. The methodology is tested on a series of cases including shock formation and flows dominated by coherent structures.

1This work is supported through a subcontract from Continuum Dynamics, Inc. under Navy STTR Phase II contract N00335-17-C-0158

10:57AM S19.00003 Slug Flow Prediction for Subsea Applications Using Dynamic Anisotropic Mesh Optimisation with Tetrahedral Control-Volume Finite Elements. CLAIRE HEANEY, LYES KAHOUDJI, LLUIS VIA-ESTREM, ASIRI OBEYSEKARA, PABLO SALINAS, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — We present a three-dimensional Direct Numerical Simulation of two-phase air-water flow inside complex pipe configurations with very large aspect ratios (Length/Diameter ¿100) for subsea applications. We focus on the challenging slug flow regimes using a dynamically unstructured mesh, which can modify and adapt to the complex air-water interface in order to represent optimally these flows. The number of unknowns of the resulting model is reduced by a mixed control-volume and finite element formulation and a volume-of-fluid method for the interface-capturing based on a compressive control-volume advection method. The resulting slug length and frequency are compared with experimental data for horizontal pipes.

Engineering and Physical Sciences Research Council UK (MUFFINS grant EP/P033180/1), PETRONAS, Royal Academy of Engineering Research Chair for OKM

11:0AM S19.00004 LES of Compressible Gas Flow Impinging on a Wall using High Order Schemes in an Unstructured Grid. DOUGLAS FONTES, MICHAEL KINZEL, University of Central Florida — In this work, compressible gas flow impinging on a wall is solved using large eddy simulation in an unstructured mesh. In this kind of flow, complex phenomena such as shock waves, high gradients, turbulence, and wall interaction can arise. In most Computational Fluid Dynamics (CFD) codes, typically with unstructured grids, spatial schemes are limited to first and second order due to the difficulties of obtaining the information of neighboring elements. This limitation results in a higher computational cost to achieve a specific accuracy. Higher-order schemes in unstructured grids, using a flux reconstruction method, have been implemented in an open source code termed as PyFR. Thus, the present effort explores the application of PyFR to perform Large Eddy Simulation (LES) for an impinging jet case discretized with an unstructured mesh. In this study, we will identify turbulent structures and the formation of instabilities. In addition, comparisons will be developed with RANS prediction. All numerical predictions will be compared to an experimental case in order to analyze the numerical accuracy in terms of experimental results.

11:23AM S19.00005 DEM and Coarse-grained modeling of bubble and particle behavior in fluidized beds. OSCAR ANTEPARA, ANN ALMGREN, MICHELE ROSSO, ROBERTO PORCU, Lawrence Berkeley National Laboratory, JORDAN MUSSER, National Energy Technology Laboratory, WILLIAM FULLMER, National Energy Technology Laboratory; Leidos Research Support Team, CHRISTOPHER BOYCE, Department of Chemical Engineering, Columbia University — MFiX-Exa is a new code being developed by the National Energy Technology Laboratory and Lawrence Berkeley National Laboratory as part of the U.S. Department of Energy’s Exascale Computing Project. MFiX-Exa originated by combining discrete element method (DEM) modules of the classic MFiX code (https://mfix.netl.doe.gov) with a low Mach number projection method for the continuous fluid phase. The new algorithm is implemented using the AMReX software for massively parallel block-structured applications (https://amrex-codes.github.io). In this work, we present the coarse-grained(CG) DEM technique in which several particles are lumped into a single Lagrangian parcel. The work focuses on comparisons, in terms of computational time and accuracy, between the traditional (single particle) DEM and the CG-DEM. The problem of interest is a cylindrical gas-solid fluidized bed containing several million 1 mm particles. Comparisons are also made to the experimental data which include bubble characteristics, bed height, and particle velocity distributions. The results assess how accurately the models can reproduce the main characteristics of the gas-solid fluidized bed and the reduction of computational time between the models.

1This research was supported by the Exascale Computing Project (17-SC-20-SC), a collaborative effort of the U.S. Department of Energy Office of Science and the National Nuclear Security Administration.

11:36AM S19.00006 Wavelet-based adaptive simulations of flapping insects. THOMAS ENGELS, LMD-CNRS, Ecole Normale Superieure-PEL, KAI SCHNEIDER, Aix-Marseille Univ, JULIUS REISS, Technische Univ, MARIE FARGE, LMD-CNRS, Ecole Normale Superieure-PEL, DMITRY KOLOMINSKYI, Japan Agency for Marine-Earth Sci and Tech (JAMSTEC) — We present a novel wavelet-based approach to compute multiscale flows generated by complex, time-dependent geometries, motivated by the spectacular flight capabilities of flying insects. Our framework is inherently based on dynamically evolving grids. Our approach is developed in a datastructure based on locally regular Cartesian blocks, which are indexed in a tree-like fashion. The blocks are distributed among MPI processes and allow an efficient parallelization for large scale supercomputers. To avoid solving elliptic problems, we approximate an incompressible fluid using the method of artificial compressibility. Since our grid is locally Cartesian, we use the volume penalization method to include moving obstacles without the need for a boundary-conformal grid. We employ biorthogonal interpolating wavelets as refinement indicators and prediction operators, and combine them with a 4th order finite difference discretization. Using thresholding of wavelet coefficients, we show that the precision of the underlying uniform discretization is maintained on our adaptive grids, while reducing the computational effort. We derive scaling relations for the numerical parameters, and present validation cases to assess its accuracy and performance on massively parallel computer architectures.

1Financial support from the ANR (Grant No. 15-CE40-0019) and DFG (Grant No. SE 824/26-1), project AIFIT, is gratefully acknowledged. The authors acknowledge HPC resources of IDRIS (No. 2018-91604) by GENCI. D.K. gratefully acknowledges financial support from the JSPS KAKENHI Grant No. JP18K13693.
Wicking plays a key role in many industrial and biological applications such as passive capillary-driven cooling technology, oil recovery, inject printing and DNA chips. Nowadays, the attempt to design an appropriate surface to enhance the wicking capabilities is getting always more significant. In fact, the control of the wetting properties has increased enormously over the past few decades. These properties are strongly related to the spreading features of a surface. Due to the advances in nanotechnology, it has been possible to tune the wettability of a surface by modifying its topography. Different works have reported the effect of the height of micro/nano structures on the wicking transport, but they have shown contrasting results. In this work, the effect of the height on Si random nanowires has been investigated. In this regard, different nanowire arrays with different heights were fabricated. Therefore, a method to characterize accurately both the wettability and wicking of super-hydrophilic surfaces has been developed and validated. After a proper characterization of the samples, it has been observed that taller nanowires provide stronger wicking.

**10:44AM S21.00002 The Spreading of Superfluid Drops**¹, MATTHEW WALLACE, DAVID MALLIN, MICHAEL MILGIE, University of California, Irvine, KENNETH LANGLEY, ANDRES AGUIRRE-PABLO, SIGUDUR THORODDSEN, King Abdullah University of Science and Technology, PETER TABOREK, University of California, Irvine — We have used video microscopy and interferometry to investigate the spreading of normal and superfluid helium drops impacting on a sapphire substrate in a saturated atmosphere of its own vapor. We find that in spite of having zero viscosity, the short-term spreading of superfluid drops (time t less than 30 ms) is nearly identical to normal helium; in both cases, the drop spreads to a characteristic diameter of 5 mm and assumes a pancake-like shape. Both normal and superfluid drops shrink with time; the normal helium drops last up to 15 minutes, but the superfluid drops last only 2-15 seconds. Superfluid drops shrink via superflow through a fluid film at the contact line, flowing at the Feynman critical velocity. Remarkably, the drops undergo a two-phase geometry-dependent retraction. During the first phase, the drop is toroidally-shaped and the radius shrinks linearly in time; in the second phase the drop assumes the shape of a spherical cap and shrinks with the square root of time. Superfluid outflow causes the drop edges to become ragged and frayed, and causes droplets of fluid that appear to form spontaneously outside the expanding drop (exodroplets.) We provide detailed maps of drop topography and contact angle.

¹King Abdullah University of Science and Technology

**10:57AM S21.00003 Simulation of Droplet Spreading Dynamics by Particle Finite Element Method Based Model and Hydrodynamic Lubrication** , ELAF MAHROUS, R. VALENY ROY, University of Delaware, ALEX JARAUTA, MARC SECANEL, University of Alberta, PAVEL RYZHAKOV, Cimne. International Centre for Numerical Method in Engineering — Modeling droplet dynamics is an active area of research. Of particular interest is the prediction of spreading rates and spatio-temporal evolutions of droplets of varying physical properties such as surface tensions on substrates with different wettability. We adopt two distinct numerical approaches: Particle finite element method (PFEM) and hydrodynamic lubrication. PFEM is advantageous due to its ability to track evolving fluid domains and provide an accurate mesh-based boundary description which facilitates the computation of the surface tension in droplet problems. An alternative modeling approach is hydrodynamic lubrication which is based on a small slope approximation, thus requiring small contact angles. The key advantage of this approach is its low computational requirements. Using a no-slip prevents any contact line movement, and applying perfect slip leads to unrealistically large velocities. A particular challenge is to relieve this singularity while accounting for dynamic contact angles. We show how this can be resolved for both methods. Also, we compare their contact line velocities with hydrodynamic analytic model, as well as with experimental results of capillary driven water droplets on substrates with widely differing properties.

**11:10AM S21.00004 Oil drop spreading on a liquid substrate** , VARUN KULKARNI, SUHAS TAMVADA, SUSHANT ANAND, University of Illinois at Chicago — Several systems in nature such as oil spills, food dressings, and pharmaceutical drugs demonstrate an interaction of oil and water. The process of merger of a single drop and the underlying bulk liquid represents one such scenario which has so far not received much attention in literature. In this study we use high speed imaging to experimentally investigate the dynamics of oil drop deformation and spreading during a gentle deposition of an oil drop onto an air-water interface. After contact with the bulk the merger of the oil drop and its spreading is found to either proceed in stages or instantly depending on the viscosity of the oil drop as characterized by its Ohnesorge number ($Oh = \mu p / (\rho p D^2)$). The topological features during transition scale as a function of the drop viscosity and are theoretically validated using appropriate force balances. It is found that the spreading behavior of the drop depends dominantly on the viscosity of the oil, ultimately determining the extent of film coverage over the liquid substrate. To validate the spreading behavior of the oils, a theoretical model based on the damped harmonic motion is also presented.

**11:23AM S21.00005 ABSTRACT WITHDRAWN —**

**11:36AM S21.00006 Water-air dynamic contact angle hysteresis on rough metal substrates**¹, MARINA MACHADO², JOSEPH MURPHY², WILLIAM RICE³, VLADIMIR ALVARADO², University of Wyoming — Ice adhesion measurements for impact ice and statically frozen water are critical for solving important aerospace challenges, such as ice-resistant coatings. The surprisingly large spread of ice adhesion values obtained on seemingly very similar materials indicate that additional considerations, such as surface roughness and contact-line spreading, need to be incorporated into the analysis of ice adhesion. In this work, we focus on the dynamics of contact angle on metal substrates of interest, such as stainless steel and aluminum, using a pendant-drop system over a frequency range 0-10 Hz. Surface roughness of metal substrates is measured using confocal microscope profilometry mapping. Dynamic contact angle hysteresis turns out to be a weak function of frequency in the range examined, but a strong function of the type, e.g., anisotropy and spatial correlation, and degree of surface roughness. These results might partially explain differences and large spread reported in the literature regarding impact ice adhesion.

¹This work was funded by NASA through the award WY-80NSS17M0049
²Department of Chemical Engineering
³Physics & Astronomy Department
⁴Physics & Astronomy Department
⁵Department of Chemical Engineering
10:57AM S22.00003 Jet atomization of brine to achieve zero liquid discharge1, CHRIS-TIAN MACHADO, YOUSHUA JIANG, KYOO-CHUL PARK, Northwestern University — Achieving zero liquid discharge in brine management facilities is critical to solving the environmental problems associated with returning high salinity water to its source. One such pathway for reducing the amount of discharged brine is by utilizing evaporation. To accelerate the evaporation of water in brine, the bulk brine solution should be atomized into microdroplets (i.e., fog) using a high pressure flow. In this study, brine with systematically varying concentrations of salt (NaCl) and surfactant (cetrimonium bromide (CTAB)) was atomized at different compressed air pressures. Results show that the rate of fog droplet generation decreases with increasing salinity. To combat this effect, adding a low surface tension surfactant and increasing compressed air pressure have shown to improve the rate of atomization. Quantitative data analysis was performed to understand the effects of the brine's physico-chemical properties, such as density, surface tension, and viscosity, as well as external variables such as compressed air pressure on overall droplet generation. This study introduces a new approach of brine evaporation using recovered thermal energy, and provides insights in jet atomization.

1Supported by the Water Collaboration Seed Funds program of the Northwestern Center for Water Research.

10:44AM S22.00002 Dynamics of impulsively induced viscoelastic jets, EMRE TURKOZ, ExxonMobil Research and Engineering, Corporate Strategic Research, HOWARD STONE, CRAIG ARNOLD, LUC DEIKE, Princeton University — Understanding the physics of viscoelastic liquid jets and their breakup is relevant to jet-based printing and deposition techniques. In this study, we study the behavior of jets induced from viscoelastic liquid films. We use the mechanical impulse provided by a laser pulse to actuate jet formation. The parameter space governing the maximum jet length and the droplet size of the resulting viscoelastic liquid jets is investigated using simulations, which are validated with experiments. To investigate the effect of viscoelasticity, we present direct numerical simulations and solve the two-phase axisymmetric momentum equations together with the volume-of-fluid technique for interface tracking and the log-conformation transformation to solve the viscoelastic constitutive equation. We show that changing the Deborah number, which denotes the ratio between the elastic and capillary time scales, can change the resulting droplet size and the maximum jet length, so that it is possible to increase the resolution of jet-based printing and deposition techniques by modifying the elasticity of the liquid material to be printed.

10:31AM S22.00001 Deformation and breakup of droplets in an oblique continuous air stream, SURENDRA KUMAR SONI, PAVAN KUMAR KIRAR, PANKAJ KOLHE, KIRTI SAHU, Indian Institute of Technology Hyderabad, India — We experimentally investigate the deformation and breakup of droplets interacting with an oblique continuous air stream. A high-speed imaging system is employed to record the trajectories and topological changes of the droplets of different liquids. The droplet size, the orientation of the air nozzle to the horizontal and fluid properties are varied to study different breakup modes. We found that droplet possessing initial momentum prior to entering the continuous air stream exhibits a variation in the required Weber number for the vibrational to the bag breakup transition with a change in the angle of the air stream. The critical Weber numbers for the bag-type breakup are obtained as a function of the Eotvos number, angle of inclination of the air stream and the Ohnesorge number. It is found that although the droplet follows a rectilinear motion initially that transforms to a curvilinear motion at later times when the droplet undergoes topological changes.

Session S22 Drops: Instability and Breakup II 604 - Larry Li, HKUST

11:49AM S21.00007 Droplet Depinning under Forcing by Wind and Gravity1, EDWARD WHITE, Texas A&M University, ALIREZA HOOSHANGINEJAD, University of Minnesota, ROGER SIMON, ELEAZAR HERRERA HERNAN-DEZ, Texas A&M University, SUNGYON LEE, University of Minnesota — Partially wetting drops are ubiquitous in nature and industry. In many cases such as rain drops on windows, droplets are subject to combined effects of gravity and wind forcing. Stability of water drops under forcing by wind and gravity is relevant to aircraft icing, heat exchangers, and fuel cells. Recent studies by Schmucker and White (2012) demonstrate a sharp transition in the trend of critical wind speed for droplet depinning from any inclined surface as a function of droplet size. To investigate what marks this self similar behavior, we conduct additional experiments of water drops on an inclined aluminum surface subject to forcing by wind. We find that the transition coincides with the onset of depinning for the advancing contact line. Inspired by this observation, we develop a mathematical model to rationalize the original experimental results of Schmucker and White.

1Supported by the National Science Foundation through Grant CBET- 1839103

10:42PM S21.00008 Impulse-driven drop, HAMED HABIBI, ROUSLAN KRECHETNIKOV, University of Alberta — Drop deformation and disintegration regimes have been studied in many contexts ranging from an impact on a solid surface or a liquid layer of varying thickness to an aerodynamic shock wave propagating in air and hitting a suspended liquid drop. As a counterpart, we study deformation and disintegration of a sessile drop on a stiff membrane in the context of an impulsive acceleration. The key objectives are here to elucidate the effects of viscosity, surface tension, and wetting on the drop deformation and disintegration as well as to convey the physical mechanisms behind the observed transient responses of the initially static drop of controlled shape and size. Hence, the significant amount of experimental data – used to map the possible drop morphological changes along with the transitions between them: crown height, radius, and instability wavelength in the crown regimes; drop detachment and disintegration times as well as probability density functions of the secondary droplets in the disintegration regimes – is interpreted with phenomenological models, scalings, and estimates highlighting the rich multiscale physics of the impulse-driven drop phenomena.

Tuesday, November 26, 2019 10:31AM - 12:02PM

10:57AM S22.00003 Jet atomization of brine to achieve zero liquid discharge1, CHRIS-TIAN MACHADO, YOUSHUA JIANG, KYOO-CHUL PARK, Northwestern University — Achieving zero liquid discharge in brine management facilities is critical to solving the environmental problems associated with returning high salinity water to its source. One such pathway for reducing the amount of discharged brine is by utilizing evaporation. To accelerate the evaporation of water in brine, the bulk brine solution should be atomized into microdroplets (i.e., fog) using a high pressure flow. In this study, brine with systematically varying concentrations of salt (NaCl) and surfactant (cetrimonium bromide (CTAB)) was atomized at different compressed air pressures. Results show that the rate of fog droplet generation decreases with increasing salinity. To combat this effect, adding a low surface tension surfactant and increasing compressed air pressure have shown to improve the rate of atomization. Quantitative data analysis was performed to understand the effects of the brine’s physico-chemical properties, such as density, surface tension, and viscosity, as well as external variables such as compressed air pressure on overall droplet generation. This study introduces a new approach of brine evaporation using recovered thermal energy, and provides insights in jet atomization.

1Supported by the Water Collaboration Seed Funds program of the Northwestern Center for Water Research.

11:10AM S22.00004 Thin-Film Breakup Dynamics of a Binary Mixture Drop during Evaporation, H. ALEX GUO, Department of Mechanical Engineering and Materials Science, Duke University, Durham, North Carolina 27708, THOMAS P. WITELSKI, Department of Mathematics, Duke University, Durham, North Carolina 27708, CHUAN-HUA CHEN, Department of Mechanical Engineering and Materials Science, Duke University, Durham, North Carolina 27708 — When a drop consisting of two volatile liquids evaporates on a solid substrate, the parent drop frequently bursts into tiny droplets. Although the bursting originates from solutal Marangoni stresses, like the wine-tear phenomenon, the binary-drop setup is distinct with its moving contact line and the dramatic bursting of the entire parent drop before complete evaporation. For a drop of water and isopropanol, a ridge develops at the contact line when the drop spreads on the silicon substrate. When evaporation drives the contact line to eventually recede, the ridge formed during spreading fragments into tiny droplets. We have developed a numerical model for the thin-film evolution of the binary drop. The model accounts for the liquids differential evaporation, which generates Marangoni stresses. Our model captures the ridge formation process, and predicts a threshold mixing ratio for the Marangoni bursting that is experimentally observed.
Evaporation-Induced Breakup of a Droplet in a Shallow Well

11:23 AM S22.00005

Kris F. WiedenhofT, H. Alex GuO, Department of Mechanical Engineering and Materials Science, Duke University, Durham, North Carolina 27708, Thomas P. Wietelski, Department of Mathematics, Duke University, Durham, North Carolina 27708, Chuan-Hua Chen, Department of Mechanical Engineering and Materials Science, Duke University, Durham, North Carolina 27708 — Droplet evaporation is most frequently studied on a flat substrate, for which surface defects are typically accounted for by contact angle hysteresis. To directly investigate the effect of surface defects, we study droplets levitated in an electrodynamic balance and the contact radius. We have developed a thin-film evaporation model that captures the annular breakup pattern as well as the range of aspect ratio achievable. The annular breakup presents a drop evaporation mode that is distinct from the widely reported modes with constant contact angle or constant curvature. To understand the effect of surface defects, we study droplet evaporation in a shallow circular well, created by etching a smooth silicon substrate. An inkjet printer is used to print water droplets onto the well, and the droplets are observed using high-speed cameras. We find that the droplet breaks in the upward direction. The experimental observations are validated with BEM (boundary element method) simulations and a reasonable agreement is observed between the two.

Thermal Atomization during Droplet Impingement on Superhydrophobic Substrates

11:36 AM S22.00006

Mohit Singh, Neha GawanDe, Y.S. Mayya, Ruchi Thakor, Indian Institute of Technology Bombay — Experimental observations are reported on the effect of the initial perturbation on the mechanism of Rayleigh breakup phenomenon of a charged droplet (diameter~100-250 μm), levitated in an electrodynamic (ED) balance. As the droplet undergoes evaporation, the droplet size decreases with a corresponding increase in the surface charge density near to the Rayleigh limit, leading to its breakup. All the successive events such as droplet deformation, breakup and relaxation of the drop after jet ejection have been captured using a high-speed camera at around 200 thousand fps. It is observed that the droplet surface exhibits finite amplitude of oscillations with higher prolate deformation on account of unbalanced gravity. These perturbations lead to subcritical Rayleigh breakup of the droplet. There exists a “x” phase shift between the centre of mass motion and applied field which causes the asymmetric breakup in such a way that the droplet breaks in the upward direction. The experimental observations are validated with BEM simulations and a reasonable agreement is observed between the two.

Thermal Atomization during Droplet Impingement on Superhydrophobic Substrates

11:49 AM S22.00007

Preston Emerson, Julie Crockett, Daniel Maynes, BYU — Water droplets impinging on superheated substrates may be characterized by dynamic droplet boiling, causing an upward ejection of miniscule secondary droplets, called droplet boiling. In this study, droplets impact superheated superhydrophobic substrates of varying microstructure configuration for a range of superheat temperatures between 120 and 320 degrees Celsius. Thermal atomization is captured using a high-speed camera and is quantified by estimating the amount of liquid sprays present for each impingement event using a 2D imaging processing technique. The start time, quantity, and velocity of the atomization sprays is shown to depend on the microstructure configuration of the substrate (most notably the height and center-to-center spacing of the structures) and the Weber number of the impinging droplet. The droplet boiling regimes are identified for the range substrate temperatures and the Leidenfrost temperature is estimated for each scenario.

Dynamic buckling of collapsing viscous bubbles

10:31 AM S23.00001

Alexandros Oratis, Boston University, John Bush, Massachusetts Institute of Technology, Howard Stone, Princeton University, James Bird, Boston University — When air bubbles rise to the surface of a liquid, they create a thin-film dome that eventually ruptures. In liquids with relatively low viscosity, film rupture is followed by rapid film retraction dominated by surface tension and inertia, and typically occurs over a period of milliseconds. In liquids with relatively high viscosity, viscous dissipation slows the film retraction sufficiently that the bubble collapses. As it does so, radial wrinkles appear on its surface. Previous investigations have concluded that gravity is responsible for both the bubble collapse and the wrinkling instability. We here demonstrate that experiments yield the same radial wrinkle pattern independent of the bubble orientation relative to gravity. We develop an alternative model for the wrinkles in which surface tension initiates a dynamic buckling instability.

Flow and mixing dynamics of phase-transforming multicomponent fluids

10:44 AM S23.00002

Hector Gomez, Saikat Mukherjee, Purdue University — We develop a continuum model for two-component flow, where one of the components is an non-condensable gas and the other one is a fluid that undergoes liquid-vapor phase transformations accompanied by changes in its miscibility with the gas. We derive the model from a Gibbs free energy that includes gradients of the fluid density and gas concentration, leading to a generalization of the classical equations of multiphase flow hydrodynamics. High-fidelity numerical simulations of the model show very complex interplay between flow, mixing and phase transformations. The model predicts quantitatively the saturation vapor pressure of water for a given mixture of air and water vapor at different temperatures. When applied to the problem of collapse of cavitation bubbles, the model shows that even very small amounts of gas dissolved in the liquid phase can have a significant impact on the dynamics of the collapsing bubble.

Bursting Bubbles and The Formation of Gas Jets and Vortex Rings

10:57 AM S23.00003

Ali Al Dasouqi, David Murphy, University of South Florida — Bursting bubbles play an important role in ocean-atmosphere and industrial processes (e.g., marine aerosol formation, froth flotation for metal concentration) and food science (e.g., beer). Earlier work focused on the fluid dynamics of droplet formation (film and jet drops) and film cap retraction, but the ejection of pressurized gas from inside a bursting bubble, which may affect the spatial distribution of generated droplets, is much less studied. Here we examine the gas flow emerging from the bursting of smoke-filled bubbles floating at an air-water interface using two synchronized orthogonal high-speed cameras filming at 5-10 kHz. We describe the bubble bursting and subsequent formation of vortex rings for bubbles ranging in size from 400 microns to 4 cm. No vortex rings are formed for bubbles less than 1 mm diameter. The number of initially formed vortices increases from one for a 1.4 mm bubble up to six for a 39 mm bubble. Using stereophotogrammetric tracking, the initial speed of the gas jet released from the bursting bubble also is quantified and can reach up to 9.6 m/s for a 39 mm bubble. Jet speed is found to be a function of film retraction speed, bubble diameter, and bubble submergence. Successful vortex ring formation also is found to rely on film retraction behavior.
11:10AM S23.00004 Collective Effects in Bursting Bubbles Aerosol Production

BAPTISTE NEEL, LUC DEIKE, Princeton University — The bursting of surface bubbles, understood as a production mechanism of sea spray aerosols, is a key feature of gas and mass transfers between ocean and atmosphere. While the case of a single bursting bubble has been extensively studied recently, little is known about collective effects in this context. Our experimental study characterizes the dynamical and statistical properties of an ensemble of initially mono-disperse air bubbles at the water surface, and the resulting spray droplets being produced. After rising in a still bath, the bubbles stand at the free surface, where they coalesce and move around before they eventually burst. The addition of surface active agents, because they prevent bubbles coalescence above a certain concentration, modify the features of the surface bubbles population (coalescence and lifetime), whose consequences on the spray production are discussed.

1This work is supported by a grant from NSF Physical Oceanography to L.D. (Grant No. 1849762.)

11:23AM S23.00005 Jet drops produced by bursting bubbles: number, size, velocity and resulting mass transfer

ALEXIS BERNY, THOMAS SEON, Sorbonne Université, CNRS, UMR 7190, Institut Jean le Rond D’Alembert, F-75005 Paris, France, LUC DEIKE, Department of Mechanical and Aerospace Engineering, Princeton University, Princeton, New Jersey 08544 USA, STEPHANE POPINET, Sorbonne Université, CNRS, UMR 7190, Institut Jean le Rond D’Alembert, F-75005 Paris, France — When a bubble bursts at a liquid-air interface, it produces a jet that may break up and eject drops called jet drops. Numerous studies focused on this phenomenon motivated by the wide range of application, from the bubble in a glass of champagne to spray generation at the surface of the ocean. Here, we solve the two-phase Navier-Stokes equations in axi-symmetrical coordinates with the free software basilisk. We first compare the size and velocity of the first drop in our simulations with the recent experimental, numerical and theoretical results from the literature, before characterizing the number, size and velocity of all ejected droplets. This approach is done for a wide range of controlling parameters, defined as the Laplace and Bond numbers. The resulting total vertical momentum and mass transfer is then discussed.

1This work was supported by the National Science Foundation under Grant No. 1849762. to L.D. and the Cooperative Institute for Earth System modeling between Princeton and GFDL NOAA. A.B. was supported by the International Fund to L.D. from Princeton University during part of this study.

11:36AM S23.00006 The physics of bubble bursting followed by jet ejection

JOSE M. LOPEZ-HERRERA, ALFONSO M. GANAN-CALVO, Universidad de Sevilla, ETSI, 41092 Sevilla, Spain — Under the light of recent research on this phenomenon, and performing exhaustive simulations, dimensional and similarity analysis, the collapse of a supercritical bubble and the subsequent ejection of a microjet are studied here in great detail. The different models proposed, their description of the physics involved, their degree of innovation and their predictive abilities comparing with experiments are put into perspective. In particular, the conditions under which the phenomenon can be described around an elusive temporal singularity, and the resulting self-similar flow configuration are carefully analyzed. Using high resolution numerical simulation techniques of free-surface flows, this study demonstrates the existence of said singularity and the value of the critical parameter for which it appears, showing the natural change in the time scaling law of the self-similar flow sufficiently close to the singularity. This self-similar flow shows a rich topology that is discussed in this work.

1Supported by the Ministerio de Economía y Competitividad (Spain), Plan Estatal Retos, project DPI2016-78887-C3-1-R

11:49AM S23.00007 Compressibility Effects for Spherical Bubble Collapse in Water

ROY BATY, SCOTT RAMSEY, CORY AHRENS, JASON ALBRIGHT, Los Alamos National Laboratory — This presentation outlines the derivation and solution of a potential flow equation used to model the effect of compressibility on bubble collapse in water. An unsteady potential flow equation is developed by adding a perturbation term to a classical incompressible flow solution describing the radial motion of a spherical bubble. The potential equation is obtained by linearizing a perturbation term about an incompressible flow field modeling the collapse of a spherical cavity in water. The potential equation describes the unsteady radial motion of a one-dimensional, inviscid, spherical bubble. The potential equation is performed for an arbitrary isotropic equation of state for water over the pressure and density range observed for cavitation. The resulting linearizing a perturbation term about an incompressible flow field modeling the collapse of a spherical cavity in water. The compressible potential equation is developed by adding a perturbation term to a classical incompressible flow solution describing the radial motion of a spherical bubble as a function of time. The flow is assumed to be one-dimensional and inviscid. Both linear and nonlinear forms of the compressible potential equation are presented assuming a general isentropic equation of state for water. The linear compressible potential flow equation is obtained by linearizing the perturbation term about the incompressible collapsing flow field. The linear and nonlinear compressible potential flow equations are integrated numerically for an analytical equation of state approximating water in the kilobar pressure range.

12:02PM S23.00008 Lie Symmetries of a Potential Flow Equation Modeling Compressibility Effects of Spherical Bubble Collapse in Water

SCOTT RAMSEY, ROY BATY, ERIC ALBRIGHT, CORY AHRENS, Los Alamos National Laboratory — This work applies analytical methods for differential equations to derive Lie symmetries associated with a potential flow equation used to model the effect of compressibility on bubble collapse in water. The compressible potential equation describes the unsteady radial motion of a one-dimensional, inviscid, spherical bubble. The potential equation is obtained by linearizing a perturbation term about an incompressible flow field modeling the collapse of a spherical cavity in water. The symmetry analysis is performed for an arbitrary isotropic equation of state for water over the pressure and density range observed for cavitation. The resulting symmetry groups obtained from the Lie analysis are then related to a general form of the method of characteristics developed to integrate second order hyperbolic equations.

Tuesday, November 26, 2019 10:31AM - 12:15PM —
Session S24 Drops: Electric Field Effects II

10:31AM S24.00001 Sorption-controlled electrohydrodynamics of a surfactant-covered viscous drop

HEREVE NGANGUIA, Indiana University of Pennsylvania, WEI-FAN HU, National Chung Hsing University, MING-CHIH LAI, National Chiao Tung University, YUAN-NAN YOUNG, New Jersey Institute of Technology — Surfactants physico-chemistry is often exploited to control the dynamics of viscous drops and bubbles. For instance, the adsorption-desorption kinetics plays a critical role in the deformation of drops in extensional and shear flows. By contrast these kinetics effects have yet to be accounted for drops in an electric field to the best of our knowledge. In this talk we present work on the effects of sorption kinetics on a surfactant-covered viscous drop in an electric field (both dc and ac). Specifically, we look at the dependence of drops deformation on three dimensionless numbers: the electric capillary number $Ca_E$, Biot number $Bi$, and Peclet number $Pe$. We present the numerical methods employed for simulations, and discuss preliminary results from our findings as well as future extensions. We also illustrate how our results may be applied to explain recent experiments on extreme drop deformation under an electric field (Brosseau and Vlahovska, PRL 2017).
10:44AM S24.00002 Electrocoalescence behavior exhibited by anchored aqueous droplets in air , RAUNAQ HASIB, ROCHISH THAOKAR, Indian Institute of Technology Bombay — Electrocoalescence of droplets is the preferred method in phase separation owing to its low energy consumption. In this work, effect of droplet conductivity on electrocoalescence phenomena is studied for an anchored droplet-in-air system under the influence of direct current electric field. Phase diagrams representing the coalescence/non-coalescence behaviour are constructed for three different droplet conductivities. Droplets of deionized water, 0.01 M NaCl solution, and 1 M NaCl solution are used for low, moderate and high conductivity experiments, respectively. Subsequent events after first non-coalescence event are also studied and an effort is made to explain and categorize the behaviour under different regimes. Beyond the critical electrocapillary number (estimated to be ~0.25), the stabilizing capillary force cannot balance the destabilizing electric force which leads to contact of droplets. Variation in drop conductivity, cone angle, and separation distance between droplets do not influence the critical electrocapillary number. Low light experiments were conducted to detect presence of sparks during the non-coalescence events. The non-coalescence mechanism is observed to differ with change in droplet conductivity.

10:57AM S24.00003 A study of non-coalescence of aqueous droplets suspended in castor oil under electric field , SUBHANKAR ROY, ROCHISH THAOKAR, Indian Institute of Technology Bombay — Experimental and numerical investigations are carried out to study the effect of electric field on coalescence and non-coalescence behaviour of two water droplets suspended in an insulating oil (castor oil). Unlike immediate breakup of the bridge, as reported in earlier studies like Ristenpart et al. (2009) [Non-coalescence of oppositely charged drops. Nature 461 (7262), 377–380], the non-coalescence behaviour observed in our experiments show that at higher than critical electric fields the droplets form a bridge which starts thickening thereby exhibiting a tendency to coalesce. However, soon this phenomenon slows and comes to a stop, and the thickened bridge starts thinning dramatically, initiating the phenomenon of eventual non-coalescence. Numerical simulations using boundary integral method are able to explain the physics behind the thickening of this bridge, followed by thinning and non-coalescence. The fundamental reason is the competing meridional and azimuthal curvatures of the bridge under the effect of electric field induced Maxwell stresses which determine the Laplace pressure inside the bridge to become either positive or negative thereby determining the direction of fluid flow towards or away from the centre of the bridge. Velocity and pressure profiles confirm this postulation, thus enabling us to predict this behaviour of transitory coalescence followed by non-coalescence.

11:07AM S24.00004 Abstract withdrawn

11:23AM S24.00005 Shaping of Charged Sprays , PAUL W. VESELY1, RUDOLF J. SCHICK, Spraying Systems Co. — Spraying by conventional hydraulic and air-assisted nozzles presents issues with overspray and uneven application. Electrostatic atomization provides an alternative method to spray oils and other electrically insulating fluids, utilizing the repulsive force of like charges instead of high fluid and air pressure. A spray plume of charged droplets provides a very high transfer efficiency when spraying onto grounded conductive substrates. A plane-to-plane electrostatic atomization nozzle produces a full cone spray plume, but for many conveyor coating processes, a flat spray is required to provide a uniform coating across a substrate. Flat plate electrodes were added outside of the electrostatic atomization nozzle near the orifice to generate an electric field that acts on the negatively charged spray plume shaping it into a flat fan plume. Various shaped and sized electrodes operating at a range of voltages of negative polarity were investigated for their effectiveness in forming the full cone spray plume, high-speed imaging, phase Doppler interferometry, and particle imaging velocimetry systems were used to investigate the effect of the external electrode configurations tested.

11:36AM S24.00006 Resolving anomalies in contrasting droplet dynamics under direct and alternating current electrical forcing, KIRTI SAHU, Indian Institute of Technology Hyderabad, SUMAN CHAKRABORTY, Indian Institute of Technology Kharagpur, India — Electrically driven dynamics of droplets has given rise to several apparent anomalies, as attributable to complex interconnections between the underlying physical forces and geometrical dimension driven morpho-dynamic topology. In sharp contrast to reported theory on electro-mechanics of droplets that trivially predicts shape oscillations of a droplet subjected to alternating current electric field about the steady state deformation under an equivalent root mean square direct current electric field under all possible electrical conductivity and permittivity contrasts of the droplet and the carrier phase, here we bring out a novel dimensionality driven physical paradigm under which the same does not necessarily hold true. Our results reveal a dramatic reversal in shape transition to an elongated profile in a direction orthogonal to the electric field, in contrast to the classically postulated elongated shape in the direction of the electric field, as the electric field is changed from alternating to a direct one, for contrasting permittivity and electrical conductivity ratios. We attribute these findings to dimensionality driven topological transitions and validate the same with reported experiments.

11:49AM S24.00007 Electrical impedance of a deforming conducting drop , PRIYA GAMHIRE, V KUMARAN, Indian Institute of Science Bangalore — Deformation of drops under electric field, both direct and alternating, has been a subject of study for nearly 5 decades now. Yet, most theoretical models describing the deformation assume fluids to be either perfect or leaky dielectrics to simplify the analysis. In a real scenario, fluids are rarely dielectric and therefore a complete theoretical description of a conducting drop deforming under an alternating electric field is lacking. Additionally, to validate the models, deformation of drops is estimated in practice via optical methods which can involve expensive piece of equipment such as high-speed cameras. If the deformation can be correlated to drop properties such as its electric impedance, it can potentially be developed into an electronic method of quantifying the deformation extending the utility of experiments. In this presentation, our attempts to find solutions to both the aforementioned points will be discussed. To describe the deformation, a Debye-Falkenhagen theoretical approach is used which describes ionic double layers. Also, an attempt made to correlate changes in the electrical impedance of the drop and its deformation will be discussed.

12:02PM S24.00008 A lattice-Boltzmann model of Electrocapillarity1 , ELFEGO RUIZ GUTIERREZ, RODRIGO LEDESMA AGUILAR, GARY G. WELLS, GLEN MCHALE, Smart Materials and Surfaces Laboratory, Northumbria University, ANDREW M. J. EDWARDS, CARL V. BROWN, MICHAEL I. NEWTON, School of Science and Technology, Nottingham Trent University — Deformation of droplets due to electric fields thus forming a bridge which connects two droplets. In contrast, the stabilizing capillary force cannot balance the destabilizing electric force which leads to contact of droplets.

1The authors acknowledge support from EPSRC Grants Nos. EP/P024408/1 and EP/R036837/1.
10:31AM S25.00001 Experimental Study of Cavitation Inception in a Pair of Interacting Vortices\textsuperscript{1}, DANIEL KNISTER, ELIZABETH CALLISON, HARISH GANESH, STEVEN CECCIO, University of Michigan - Ann Arbor — Cavitation inception in shear flows often occurs in secondary stream-wise vortices (braids) stretched by spanwise vortices. Stretching of the weaker secondary stream-wise vortices can lead to a rapid drop in core pressure below the vapor pressure, and thus inception of captured cavitation nuclei in the stretching core. Understanding of the relationship between the stretching process, pressure drop, and nuclei size is critical for understanding inception. As a model experiment of this phenomenon, two parallel vortices are created by a pair of hydrofoils in a re-circulating water channel following the study of Chang et al. (2012). Cavitation inception and occurrence caused due to interaction between two secondary hydrofoils is studied using high speed video and synchronized hydrophone measurements, with the vortices visualized by cavitation and dye injection and SPIV. We will discuss the hydrodynamic performance of the new configuration, including the parameters that lead to the desired vortex interactions and inception in the weaker (secondary) vortex.

\textsuperscript{1}This work was supported by Office of Naval Research, under program manager Dr. Ki-Han Kim, MURI grant number N00014-17-1-2676

10:44AM S25.00002 Shock-cavity interactions in a shock-compressed polymer medium in the fluid regime\textsuperscript{2}, EMILIO ESCAURIZA, University of Oxford, NIRMAL RAI, University of Iowa, DAVID CHAPMAN, University of Oxford, JOAO PEDRO DUARTE, Imperial College London, LUKASZ FARBANIEC, LIAM SMITH, JOHN JOHNSSON, MICHAEL RUTHERFORD, University of Oxford, MARGIE OLBINADO, ALEXANDER RACK, ESFR, H S UDAYKUMAR, University of Iowa, DANIEL EAKINS, University of Oxford, UNIVERSITY OF OXFORD TEAM, UNIVERSITY OF IOWA TEAM, ESFR TEAM — The study of shock-cavity interactions is important for a wide range of applications, from the medical sciences to the development of mixing mechanisms. However, due to constraints posed by optical imaging, observing the phenomenon directly has proven challenging. We present observations of shock-induced collapse in a solid medium through ultra-high-speed radiography, performed at the ESRF synchrotron. The radiography allowed the tracking of the time evolution of sub-surface interfaces during the collapse process. Shock loading of the PMMA cavity targets was achieved through plate impact with a 2-stage gas gun. As the shock strength was well in excess of the yield strength of the solid medium, the collapse shape was typical of a cavity collapsing in a fluid, with the generation of a jet and the formation of toroidal vortices after jet impact against the far cavity interface. The dependence of the collapse time on shock pressure was investigated, revealing a power law relationship. The results showcase the capabilities of high-speed synchrotron radiography for observing sub-surface phenomena in liquid and solid media.

10:57AM S25.00003 Analysis of the mechanisms of cavitation erosion\textsuperscript{3}, BEN ZHAO, OLIVIER COUTIER-DELGOSHA, Virginia Tech — Erosion related to the collapse of a single cavitation bubble is investigated. The bubble is created using a pulsed high intensity laser focused in a water tank, and the mechanisms of erosion are studied with high speed visualizations, a small time response hydrophone, and a pressure sensor mounted on the material. The effects of the distance of the bubble to the wall, the bubble size, and the softness of the material are investigated. A specific attention is paid to the feedback on the material on the bubble collapse, depending on the material stiffness. First results based on the use of two lasers creating two interacting bubbles will be also presented.

11:10AM S25.00004 Temperature measurements in cavitation bubbles\textsuperscript{4}, MEROUANE HAMDJI, Arts et Metiers ParisTech, OLIVIER COUTIER-DELGOSHA, Virginia Tech, MICHAEL BAUDOIN, University of Lille — The present work focuses on the analysis of the extreme conditions encountered during the process of collapse of cavitation bubbles. The objective is to characterize the temperature variations inside the vapor/gas bubble, and also in the surrounding liquid. The work is based on an experimental approach where temperature measurements are performed with a fast response cold wire thermometer. Specific thin cold wires obtained by Nickel metallic coating, whose resistance varies according to the local temperature, have been developed. They have been applied to configurations of single bubbles created by a travelling pressure wave. In most of the tests, the temperature peak magnitude measured at the end of the collapse was typical of a cavity collapsing in a fluid, with the generation of a jet and the formation of toroidal vortices after jet impact against the far cavity interface. The dependence of the collapse time on shock pressure was investigated, revealing a power law relationship. The results showcase the capabilities of high-speed synchrotron radiography for observing sub-surface phenomena in liquid and solid media.

11:29AM S25.00005 Inertial cavitation threshold in a viscoelastic medium\textsuperscript{5}, KAZUYA MURAKAMI, ERIC JOHNSON, University of Michigan — Cavitation bubbles play a significant role in medicine such as histotripsy or traumatic brain injury. Elevated temperatures, high strain rates, and shock waves are generated by the bubble dynamics, which may damage the surrounding tissue. These phenomena are thought to originate from inertially dominated bubble oscillations. It is thus important to determine a criterion governing inertial cavitation in tissue. For this reason, we numerically investigate the threshold for inertial cavitation in a tissue-mimicking, viscoelastic medium. We use a Rayleigh-Plesset-type equation, where compressibility, heat diffusion, mass transfer and nonlinear elasticity are taken into account. We apply a negative pressure pulse to a bubble and examine its expansion and subsequent collapse. In particular, we investigate the amount of energy dissipated by various means and determine the conditions under which inertia becomes dominant. Since viscoelasticity reduces bubble growth, the inertial cavitation threshold becomes higher in tissue-like media.

\textsuperscript{5}This work was supported in part by ONR grant N00014-17-1-2058 (under Dr. T. Bentley) and NSF grant CBET 1253157. KM acknowledges the Overseas Scholarship from the Puni Foundation for Information Technology

11:36AM S25.00006 ABSTRACT WITHDRAWN —

11:49AM S25.00007 The promotion effect of polymer nanoparticles on laser-induced thermal cavitation\textsuperscript{6}, MAN HU, FENG WANG, DAOSHENG DENG, Fudan University, Shanghai, China — Laser impacting on liquids, ranging from a single droplet to soft biological tissues, can generate cavitation. By immersing nanoparticles into water, cavitation can be manipulated. Here, we report CO\textsubscript{2} laser impacting on distilled water with polymer nanoparticles immersed in to produce thermal cavitation at the air/water interface. The promotion effect of nanoparticles on the inception of cavitation is investigated. With proper concentration nanoparticles being introduced in, thermal cavitation is clearly observed at the air/water interface from high speed imaging, but no cavitation for water with no nanoparticles. Based on this phenomenon, three regimes (no cavitation, cavitation, and pseudo-cavitation) are identified within a broad range of nanoparticles concentration and size. Moreover, this interfacial cavitation allows the direct visualization of spatial-temporal evolution of temperature, which reveals that the polymer nanoparticles not only act as preexistent nuclei to promote nucleation for cavitation, but also likely affect temperature to change the nucleation rate as well.

\textsuperscript{6}This work was supported by the Funai Foundation for Information Technology.
Tuesday, November 26, 2019 10:31AM - 12:15PM – Session S26 Flow Control: Plasma Actuation B 608 - Jesse Little, University of Arizona

10:31AM S26.00001 Active flow control of the laminar separation bubble on a plunging airfoil near stall1. JESSE LITTLE, MARK AGATE, ARTH PANDE, University of Arizona — The effects of small amplitude, high frequency plunging motion on the X-56A airfoil are examined experimentally at Re=200,000 for 12 degrees angle of attack. The purpose of this research is to study the aerodynamic influence of structural motion when the wing is vibrating close to its eigenfrequency near static stall. Specific focus is placed on the laminar separation bubble (LSB) near the leading edge and its control via plasma actuation. In the baseline case, the leading edge bubble bursts during the oscillation cycle causing moment stall. A collaborative computational effort has shown that small amplitude forcing at a frequency that is most amplified by the primary instability of the LSB generates coherent spanwise vortices that entrain freestream momentum, thus reducing separation all while maintaining a laminar flow state. Results (PIV and surface pressure) indicate that the most effective control for the lift increase can be achieved for the case that the streamwise vortex generated by the VG-PA collides with the opposite streamwise vortex. In addition, from the results of 0.05 \( c \), 0.1 \( c \), 0.2 \( c \), 0.3 \( c \), and 0.4 \( c \), where \( c \) means the chord length. The maximum lift is attained in the case with 0.1\( c \). On the other hand, a streamwise vortex generated by the VG-PA in a boundary layer flow has been investigated to clarify the distribution of the vortex. The streamwise vortex is distributed from the actuator to 0.1\( c \) away from the actuator in the induced flow direction. It means that the most effective control for the lift increase can be achieved for the case that the streamwise vortex generated by the VG-PA collides with the opposite streamwise vortex. In addition, from the results of 0.05\( c \) and 0.1\( c \), the location of the streamwise vortex collision is also important factor for the effective control.

1Supported by AFOSR

10:44AM S26.00002 Viscous Drag Reduction on a NACA 63012A Airfoil1. KATHERINE YATES2, ALAN DUONG3, THOMAS CORKE3, FLINT THOMAS4, University of Notre Dame — A series of wind tunnel experiments were performed in which an array of flush mounted pulsed-DC plasma actuators were utilized to reduce the skin friction drag on a NACA 63012A airfoil over a Mach number range of 0.20 \( \leq M_{\infty} \leq 0.50 \) at zero angle of attack. The array of plasma actuators were designed to inhibit the lift-up and subsequent break-up of the low-speed wall streak structure to prevent the formation of streamwise vortices; a key element in wall-bounded turbulence generation. Experiments were performed with two sets of actuator arrays: 1) with the electrodes aligned in the mean flow direction and 2) with the electrodes oriented 5 degrees offset to the oncoming flow. The aerodynamic load (viscous drag) was measured directly using an integrated floating element force balance. Viscous drag reduction of up to 47\% was observed depending on the operating parameters of the plasma actuators. Net power savings were also achieved across the range of Mach numbers tested.

1Supported by DARPA Phase II D17PC00073
2Student Member
3Student Member
4APS Fellow
5APS Associate Fellow

10:57AM S26.00003 Active Turbulent Boundary Layer Drag Reduction using Pulsed-DC DBD Plasma Actuators1. ALAN DUONG2, THOMAS CORKE3, FLINT THOMAS4, University of Notre Dame — Experiments were performed involving the use of an active flow control scheme designed to inhibit the lift-up and subsequent break-up of the low-speed wall streak structure to reduce skin friction drag in a turbulent boundary layer. The flow control utilized an array of pulsed-DC plasma actuators that was designed to produce a steady span-wise velocity component on the order of \( u_c \) in order to reduce the mean flow distortion caused by the quasi-steady wall streak structure first observed by Kline et al (1967). This flow control method has been successful in reducing the viscous drag over a decade of Mach numbers and is capable of reducing the skin friction coefficient up to 68\% while maintaining net power savings. The work presented here investigates the underlying flow physics of a pulsed-DC drag reduced boundary layer in a controlled, low-speed environment. The plasma actuator array in this paper was successful in reducing the skin friction velocity by 37\%, which corresponds to a decrease of 50\% in the skin friction coefficient, measured directly by means of a floating element force balance. Detailed two-component velocity measurements were done with xwire hotwires in the wallnormal and spanwise directions downstream of the actuator.

1Supported by NASA SBIR Phase II NNX16CL27C
2APS Student Member
3APS Fellow
4APS Associate Fellow

11:10AM S26.00004 On spatial arrangement of vortex-generator type plasma actuator for separation control of airfoil flow. MAKOTO SATO, CHINATSU KOBAYASHI, Kogakuin University — LESs on separation control flows around a NACA4418 have been conducted. In the separation control, a vortex-generator type plasma actuator (VG-PA) has been adapted. The flow conditions and the configurations of the plasma actuator are set based on the previous experimental study. The Reynolds number is 85000 with angle of attack 18 deg. The effects of spanwise spacing of the VG-PA have been mainly investigated. The spacing of the VG-PA is set as 0.05c, 0.1c, 0.2c, 0.3c, and 0.4c, where c means the chord length. The maximum lift is attained in the case with 0.1c. On the other hand, a streamwise vortex generated by the VG-PA in a boundary layer flow has been investigated to clarify the distribution of the vortex. The streamwise vortex is distributed from the actuator to 0.1c away from the actuator in the induced flow direction. It means that the most effective control for the lift increase can be achieved for the case that the streamwise vortex generated by the VG-PA collides with the opposite streamwise vortex. In addition, from the results of 0.05c and 0.1c, the location of the streamwise vortex collision is also important factor for the effective control.
11:23AM S26.00005 Effects of the Layout of DBD Plasma Actuators on its Anti-/De-Icing Performance for Aircraft Icing Mitigation\textsuperscript{1}, CEM KOLBAKIR, HAIYANG HU, YANG LIU, HUI HU, Iowa State University — An experimental study was performed to evaluate the effects of different layouts of DBD plasma actuators on their anti-/de-icing performances for aircraft icing mitigations. An array of DBD plasma actuators were designed and embedded on the surface of a NACA0012 airfoil/wing model in different layout configurations (i.e., different alignment directions of the plasm actuators (e.g., spanwise vs. streamwise), width of the exposed electrodes and the gap between the electrodes). The experimental study was carried out in the Icing Research Tunnel available at Iowa State University (i.e., ISU-IRT). While the dynamic anti-icing operation is recorded by using a high-resolution imaging system, a high-speed Infrared (IR) thermal imaging camera is used to quantitatively map the temperature distributions over the surface of the airfoil model during the anti-/deicing processes. The findings derived from the present study are very helpful to explore/optimize design paradigms for the development of novel plasma-based anti-/de-icing strategies tailored specifically for aircraft inflight icing mitigation to ensure safer and more efficient aircraft operation in atmospheric icing conditions. /abstract- Cem Kolbakir, Haiyang Hu, Yang Li

\textsuperscript{1}The research work is supported by National Science Foundation under award numbers of OISE1826978, CBET- 1916380 and CMMI-18248400 and Iowa Energy Center of the State of Iowa

11:36AM S26.00006 Transient flow control effect of Quasi-DC filamentary plasma in Mach 2 and Mach 4 supersonic flows\textsuperscript{1}, YASUMASA WATANABE, University of Notre Dame, The University of Tokyo, SKYE ELLIOTT, ALEC HOUGHT, SERGEY LEONOV, University of Notre Dame — This study explores the effect of pulse-periodic quasi-DC filamentary plasma on the flow structure over 15-degree compression ramp in Mach 2 and Mach 4 airflows. A major attention is focused on transient phenomena related to plasma-flow interaction. Experiments were conducted in supersonic wind tunnel SBR-50 at the University of Notre Dame. Pulse-periodic plasma generator, operating at the frequency ranging from 250 to 5000 Hz, was installed flush-mounted crossflow in front of the compression ramp. Flow stagnation pressure and temperature were varied as 1-4 bar and 300-600K respectively. The transient flow structure and plasma filament behavior were visualized with schlieren method and high speed camera. The surface pressure distribution on the wall surface and at the ramp was measured with fast pressure transducers. The plasma generated upstream of the compression ramp shifts the shock position from the ramp to the electrode location, subsequently forming a separation zone and resulting in pressure change behind electrodes and over the ramp surface both in cases of Mach 2 and Mach 4 flows. Pressure change was characterized as a function of flow/plasma parameters.

\textsuperscript{1}This work is supported by JSPS (18K13923) and by FlowPAC Institute, University of Notre Dame.

11:49AM S26.00007 Numerical, Experimental and Analytical Investigation of the Planar Electrohydrodynamic Wall Jet, PATRICK FILLINGHAM, YIFEI GUAN, RAVI SANKAR VADI, IGOR NOVOS-SELOV, University of Washington — Classical laminar wall jet theory is used for the examination of the flow induced by Electrohydrodynamic (EHD) actuators. The planar EHD wall jet, induced by both corona discharge and dielectric barrier discharge (DBD), is investigated experimentally and computationally. The thrust generated by the actuators is measured while the velocity profiles downstream of the cathode are measured using hot-wire anemometry. A multiphysics computational model couples the Navier-Stokes, electrical field, and ion transport equations. Experimental investigations of EHD actuators in the literature are used for model validation. The wall jet resulting from both DBD and Corona actuators was found to adhere to the analytical solution for a planar laminar wall jets, allowing for the calculation of the momentum generated by EHD actuators with only information about the downstream maximum velocity. The model allows for characterization of the thrust generated by any EHD actuator attached to a wall using only two velocity measurements.

12:02PM S26.00008 Drag reduction on a three-dimensional model vehicle by a wire-to-plate DBD plasma actuator\textsuperscript{1}, DONGRI KIM, ZHI WU, HYUNGROK DO, HAECHEON CHOI, Seoul National University — We apply a wire-to-plate DBD plasma actuator to a three-dimensional model vehicle (Ahmed body) for drag reduction. With a thin wire (diameter of 11 \( \mu \text{m} \)) as an exposed electrode, the performance and efficiency enhance far more than those of the conventional plate-to-plate actuator. By varying the actuator length in the spanwise direction, the drag reduction up to 10\% is obtained at the Reynolds number of 96,000 based on the free-stream velocity and model height. With surface-pressure and PIV measurements, it is shown that drag reduction occurs mainly due to entrainment induced by streamwise momentum addition and flow changes at the actuator ends, which significantly modify the evolution of the flow in the center region and corner vortices at the lateral sides of the slanted surface of the model. Other configurations of plasma actuator are being tested and their results will be also given in the presentation.

\textsuperscript{1}This research was supported by the National Research Foundation through the Ministry of Science and ICT (no. 2017R1A4A1015523).


10:31AM S27.00001 Lagrangian description of the unsteady flow induced by a jellyfish, JIN-TAE KIM, LEONARDO P CHAMORRO, University of Illinois at Urbana-Champaign — The unsteady flow induced by a single pulse of Aurelia aurita was quantified via 3D particle tracking velocimetry. Inspection of the flow included Lagrangian statistics, velocity and acceleration probability density functions (PDF), acceleration variance as well as pair dispersion. PDF of the Lagrangian velocity components indicated more intense mixing in the radial direction and revealed three stages dominated by flow acceleration, mixing, and dissipation. During the mixing phase, the flow shares characteristics of homogeneous isotropic turbulence. We show that a single pulse may induce rich dynamics, where pair dispersion exhibits a super-diffusive \( t^{1.5} \) regime during the accelerated flow due to large-scale flow inhomogeneity; this is followed by a coherent \( t^{2} \)-Batchelor scaling in the mixed wake and then \( t^{3/2} \)-Brownian motions in a late stage dominated by flow dissipation. Kolmogorov microscales during the fully mixed phase were obtained with three distinct approaches, namely, Heisenberg-Yaglom relation of the Lagrangian acceleration variance, the fluctuating rate of the strain tensor in the Eulerian frame and the Batchelor scaling in pair dispersion, which showed good agreement.
10:44AM S27.00002 Metabolic costs of enhancing propulsion in artificially controlled live jellyfish†, NICOLE XU, JOHN DABIRI, Stanford University — Artificial control of animal locomotion has the potential to address previously inaccessible questions about the biology of swimming organisms and animal-fluid interactions, where we are otherwise limited to observations of natural behavior. This work presents a biohybrid robot that uses a self-contained microelectronic system to induce swimming in live jellyfish. By driving body contractions at an optimal frequency range faster than observed in natural behavior, swimming speed can increase nearly threefold, with only a twofold increase in cost of transport to the animal. Robotic control was also used to characterize the metabolic response of the jellyfish to swimming in the enhanced mode, and it was determined that the animals can sustainably support the associated metabolic costs. These experimental results are consistent with an adapted hydrodynamic model developed to characterize enhanced propulsion. This capability can potentially be leveraged in applications such as ocean monitoring, and to enable further studies of swimming organisms in more user-controlled, systematic experiments.

†NSF GRFP

10:57AM S27.00003 Swimming via size-change: High efficiency propulsion using resonant fluid-structure interactions, GABRIEL WEYMOUTH, Gabriel Weymouth — Cephalopods use large-scale structural deformation to propel themselves underwater, changing their internal volume by 20-50%. In this work, the hydroelastic response of a swimmer comprised of a fluid-filled elastic-membrane is studied via analytic, numerical, and experimental methods. The self-propelled soft-body fluid and solid dynamics are shown to benefit greatly from the jet flow, the internal added-mass variation, and the pulsation in tune with the swimmer’s immersed fundamental frequency. It is shown that even a simplistic size-changing structure can utilize these physical mechanisms to achieve quasi-propulsive power ratios of greater than 100%, i.e. self-propulsion for these swimmers requires less energy than towing at the same speed.

11:10AM S27.00004 Validating Improved Efficiency of Bioinspired Unsteady Jetting Propulsion†, MICHAEL KRIEG, University of Hawaii, KAMRAN MOHSENI, University of Florida — Jetting propulsion has historically been considered inefficient, as the rate of momentum transfer for a continuous jet scales with the velocity squared; whereas, the rate of kinetic energy scales with the velocity cubed. For steady jets, efficiency decreases with the ratio of jet velocity to vehicle velocity. Several animals propel themselves with high velocity jets, but none jet continuously. They pause between jetting to refill, and expel the next jet starting from rest resulting in a leading vortex ring. Vortex ring formation induces a converging radial velocity increasing hydrodynamic impulse and increasing cavity pressure. Also, fluid acceleration generates propulsion without significant wake energy. This study validates improved propulsive efficiency on a freely swimming autonomous underwater vehicle (AUV). We have developed AUVs that use such thrusters for maneuvering, and previously validated propulsive efficiency measurement using motion capture position data and motor frequency data. But in a maneuvering configuration the thrusters have lower propulsive efficiency due to losses from vehicle drag. With a streamlined AUV, we demonstrate that propulsive efficiency of unsteady jetting rivals that of unducted propellers, and is nearly double the efficiency in a maneuvering configuration.

†ONR and NSF

11:23AM S27.00005 Yaw Turning Experiments of a Bio-Inspired Vessel with Undulating Fin Propulsion†, MOHAMMAD I UDDIN, GONZALO A. GARCIA, OSCAR M. CURET, Florida Atlantic University — Navigation of autonomous underwater vehicles (AUVs) in tight spaces, coastal zones and close to submerge structures remains a challenge. One of the problems preventing AUVs to navigate in these complex environments is an adequate propulsion system that allows the vessel to move in multiple directions and/or perform precise station-keeping. We present an underwater vehicle equipped with a bio-inspired fin propulsion. The propulsion system is a single flexible undulating fin that runs along the length of the robot which control forward and directional maneuvers. We establish a dynamic and control model relating different fin kinematics to performance in yaw turning maneuvers. Turning performance were tested during free-swimming experiments and compared with a numerical model. In particular, fin kinematics for two turning characteristic were considered: heading change or correction and minimum radius turns. In addition, the flow generated by the fin for turning kinematics were measured using particle image velocimetry. These experiments will be useful to establish optimal combination of the yaw turning parameters of undulating fin propulsion.

†This material is based upon work supported by the National Science Foundation under Grant No 1751548

11:36AM S27.00006 Maneuver Control of an Undulating-Fin Underwater Vessel with a Central Pattern Generator†, ALBERT ESPINOZA, Universidad Ana C. Mendez - Gurabo, GONZALO GARCIA, OSCAR CURET, Florida Atlantic University — Undulating fin propulsion for underwater vehicles provides key advantages over traditional propeller-based methods, including increased maneuverability and high efficiency at low speed. However, some of the challenges of controlling the motion of the vessel using an undulating fin propulsion are the high coupling of the propulsive forces and torques, and the extensive parameter space of the propulsive surface. In this work, we implemented a Central Pattern Generator (CPG) to control the fin and provide a smooth transition between rapidly-changing fin wave control commands. We developed a numerical model of an underwater vehicle propelled by a single undulating fin equipped with a central pattern generator to control the swimming motion and perform different maneuvers. The model includes a 6 degree-of-freedom motion of the vessel and a discretized hydrodynamic model of the fin. This model was used to study the effects of CPG dynamics on vessel response for different swimming modes, including straight line and forward-reverse motion. The results were compared to experiments using a robotic underwater vehicle.

†This material is based upon work supported by the National Science Foundation under Grant No 1751548
11:49AM S27.00007 Reinforcement Learning for a Bio-Inspired Vehicle with Undulating Fin Propulsion.1, GONZALO GARCIA, MOHAMMAD UDDIN, SIDDHARTHA VERMA, OSCAR CURET, Florida Atlantic University — Undulating fins provide natural swimmers with fascinating locomotion capabilities. However, the use of undulating fin propulsion for underwater vessel to perform specific maneuvers is non-trivial. Currently, researchers implement ad-hoc fin kinematics such as sinusoidal traveling waves due to ease of implementation, or use kinematics based on live animals. In the proposed work, we integrate reinforcement learning with simulations and underwater robotic experiments to determine optimal undulating fin kinematics for a variety of swimming performance. A six degree-of-freedom numerical model is used to simulate the motion of a vessel with an undulating fin. The swimmer is modeled as a rigid body with a single fin running along the length of the body. A reinforcement learning algorithm was developed to determine the optimal kinematics of the fin for basic locomotion, including straight swimming and turning. We find that a model-free self-learning approach can be used to generate more complex actuation combination for improved performance. The simulation results are compared to the performance of a bio-inspired vehicle with an undulating fin propulsion. The results indicate that the use of reinforcement learning could be particularly useful for unsupervised decision-making, especially in the presence of unpredictable disturbances or flow conditions.

1This material is based upon work supported by the National Science Foundation under Grant No 1751548

12:02PM S27.00008 Experiments with Impulsive Motion of a Foil to Generate Large Lift and Thrust Forces, MIRANDA KOTIDIS, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — As underwater vehicles become increasingly versatile and capable, bio-inspired propulsion systems are becoming a viable possibility for future vehicles. In particular, flapping foil actuators are promising in their abilities for propulsion and maneuvering. Current underwater vehicles rely on propellers, which form a jet wake to produce propulsion forces, and as such, experience an inherent delay between the movement of the propeller and the vehicle feeling a propulsive force. To mitigate this shortcoming, flapping foils were moved in swift, one-time strokes to produce large, transient forces in still water. These strokes take advantage of added mass/inertial effects to produce propulsive forces almost instantaneously. Various combinations of heave and pitch motions were tested and dye visualization was performed with a custom wing to elucidate the wake and vortical structures produced by these strokes.

Tuesday, November 26, 2019 10:31AM - 12:15PM – Session S28 Particle Laden Flows: Particle Resolved Simulations

10:31AM S28.00001 Cohesive Sediment in Turbulence, KUNPENG ZHAO, FLORIAN POMES, RAPHAEL OUILLON, THOMAS KOELLNER, BERNHARD VOWINCKEL, ECKART MEIBURG, UC Santa Barbara — We investigate the balance between flocculation and break-up of cohesive particles in turbulent flows by means of grain-resolving direct numerical simulations. As a first step, we consider the model problem of inertial particles moving in a steady-state, cellular flow field consisting of counterrotating vortices. The dynamics of these particles are characterized by their Stokes number and Cohesion number, as well as by the ratio of their diameter to the vortex size. These one-way coupled simulations provide information on the competition between hydrodynamic, cohesive and collision forces, the equilibrium floc size distribution, and on the time scale of the floc formation process. We find that the equilibrium floc size grows with the Cohesion number, and that flocculation progresses most rapidly for a suitably defined Stokes number near unity. In a subsequent step, we explore how these findings translate to cohesive particles moving in homogeneous isotropic turbulence.

10:44AM S28.00002 Interface-resolved simulations of small inertial particles in a turbulent channel flow1, FRANCESCO PICANO, University of Padova, PEDRO COSTA, LUCA BRANDT, KTH Mechanics — Turbulent flows laden with small inertial particles are found in different contexts. Dealing with very dilute conditions, the so-called one-way coupling regime takes place with particles transported by the fluid without back and mutual reactions. Even in this regime, models for particle dynamics are crucial to accurately simulate their transport. In this work, we compare data from interface-resolved and one-way-coupled point-particle direct numerical simulations (DNS) of a turbulent channel flow laden with small inertial particles, with high particle-to-fluid density ratio of 100 and particle diameter of 3 viscous units. The most dilute flow considered, solid volume fraction \(O(10^{-5})\) shows the particle feedback on the flow to be negligible, whereas differences with respect to the unladen case are found for volume fraction \(O(10^{-4})\). The most dilute case is taken as the benchmark for accessing the validity of usual point-particle model considering only a non-linear drag. In the bulk of the channel, particle velocity statistics from the point-particle DNS agree well with those from the interface-resolved DNS, while major differences are found close to the wall. We show that they are due to particle-wall interactions that are not reproduced by usual point-particle model.

1This work was supported by the European Research Council grant no. ERC-2013-CoG- 616186, TRITOS. We acknowledge computer time provided by SNIC (Swedish National Infrastructure for Computing), and PRACE for awarding us access to the supercomputer Marconi, based in Italy at CINECA under project 2017174185 DILPART

10:57AM S28.00003 Consolidation of freshly deposited cohesive and noncohesive sediment: Particle-resolved simulations, ECKART MEIBURG, BERNARD VOWINCKEL, EDWARD BIEGERT, PAOLO LUZZATTO-FEGIZ, UC Santa Barbara — We analyze the consolidation of freshly deposited cohesive and noncohesive sediment by means of particle-resolved direct Navier-Stokes simulations based on the immersed boundary method. The computational model is parametrized by material properties and does not involve any arbitrary calibrations. We obtain the stress balance of the fluid-particle mixture from first principles and link it to the classical effective stress concept. The detailed data sets obtained from our simulations allow us to evaluate all terms of the derived stress balance. We compare the setting of cohesive sediment to its noncohesive counterpart, which corresponds to the settling of the individual primary particles. The simulation results yield a complete parametrization of the Gibson equation, which has been the method of choice to analyze self-weight consolidation.
11:10AM S28.00004 Particle resolved simulations of a sphere settling in simple shear flows of yield-stress fluids\textsuperscript{1}, MOHAMMAD SARABJIAN, Ohio university, MARCO E. ROSTI, LUCA BRANDT, KTH Mechanics, SARAH HORMOZI, Ohio university — We perform 3D numerical simulations to investigate the sedimentation of a single sphere in the absence and presence of a simple cross shear flow in a yield-stress fluid. In our simulations the settling flow is considered to be the primary flow, whereas the linear cross shear flow is a secondary flow. To study the effects of elasticity and plasticity of the carrying fluid on the sphere drag as well as the flow dynamics, the fluid is modeled using the elastoviscoplastic (EVP) constitutive laws proposed by Saramito. We find that the drag on the sphere settling in the absence or the presence of cross flow is an increasing function of material plasticity at constant elasticity, while it is a decreasing function of material elasticity at constant plasticity. The present methodology handles general boundary conditions for the scalars. Dirichlet (Neumann) for infinitely fast (finite rate) surface reactions, and coupled heat and mass transfer description between the solid and fluid phases. A comparison with the inviscid case is also made to quantify the impact of density gradients.

\textsuperscript{1}National Science Foundation, American Chemical Society Petroleum Research, European Research Council

11:23AM S28.00005 A penalization method for DNS of weakly compressible reacting gas-solid flows\textsuperscript{1}, BAPTISTE HARDY, JURAY DE WILDE, GREGOIRE WINCKELMANS, Université catholique de Louvain — Gas-solid flows are encountered in many environmental and industrial phenomena. Simulating such flows at large scales requires closure models for interfacial mass, momentum and heat transfer. Particle-resolved simulations can support the development of improved closure laws, from first principles. The present study combines a penalization method to account for the solid phase with a weakly compressible approximation for the gas phase. Strong thermal effects from chemical reactions in the solid phase can induce significant density gradients near the particles and affect interfacial transfer laws. The present methodology handles general boundary conditions for the scalars. Dirichlet (Neumann) for infinitely fast (finite rate) surface reactions, and coupled heat and mass transfer description between the solid and fluid phases. A comparison with the inviscible case is also made to quantify the impact of density gradients.

\textsuperscript{1}Baptiste Hardy is a research fellow of the P.N.R.S-Fonds de la Recherche Scientifique, under grant n 1.A.700.18F. Computational resources have been provided by the Consortium des Equipements de Calcul Intensif (CECI), funded by the Fonds de la Recherche Scientifique de Belgique (F.R.S.-FNRS) under Grant No. 2.5020.11 and by the Walloon Region

11:36AM S28.00006 Near-wall and collision dynamics of particles at a stagnation point on a wall\textsuperscript{1}, QING LI, MICHELÈNE ABBAS, Laboratoire de Génie Chimique, Toulouse, France, JEFFREY F. MORRIS, Levich institute, City College of New York, ERIC CLIMENT, Institut de Mécanique des Fluides de Toulouse, France, JACQUES MAGNAUDET, (Institut de Mécanique des Fluides de Toulouse, France) — We present highly resolved immersed boundary simulations of neutrally-buoyant sphere (radius $a$) motion in axisymmetric stagnation (Hiemenz) flow at a wall. Far from the wall, the particle behaves as a tracer, decelerated by the ambient pressure of the Hiemenz flow. Near the wall, slip velocity and excess hydrodynamic force (in addition to ambient pressure), $F_{slip}$ play a role. Inertia is characterized by $a/\delta$ for boundary layer thickness, $\delta$. At $F_{slip}/F_{drag} \approx 2$ to 10 transitions at $a/\delta = 2$ at the wall due lubrication. The particle reaches $O(10^{-4})$ separations with $O(1)$ velocity, motivating a model for contact and rebound. Flow and collision are studied for one and two particles, with single particle motion dominated by lubrication pressure and hydrodynamic drag (latter toward the wall). For two identical particles on the axis, certain separations lead to particle collision before the lower (closer to wall) particle hits the wall; the resulting momentum exchange leads to larger impact velocity than for one particle. Dynamics of the colliding pair includes rebound without contact with the wall for the lower of two particles, due to lubrication from the upper particle from drag allowing the pressure force to dominate.

\textsuperscript{1}The authors thank financial support of NEMESIS project

11:49AM S28.00007 Rigid particle-laden flows computations with a distributed Lagrange multipliers/fictitious-domain method on an adaptive quad/oct-tree grid. CAN SELCUK, Department of Mathematics, University of British Columbia, STEPHANE POPINET, Institut Jean le Rond d’Alembert Université Pierre et Marie Curie, ANTHONY WACHS, Department of Mathematics, department of Chemical & Biological Engineering, University of British Columbia — Modeling rigid particle-laden flows requires an accurate description of the flow field in the close vicinity of the particles (i.e. in the boundary layers). One of the numerical difficulties lies on the extremely fine grid necessary to fully capture the flow dynamics near the particle boundary (as e.g. lubrication force). In such scenario, fixed Cartesian grids are prohibitive as the required number of computational cells becomes impractical. To overcome this difficulty, we combine an adaptive mesh refinement (AMR) technique with a distributed Lagrange multipliers/fictitious-domain method (DLM/FD) (Glowinski et al. 1999). The solver is implemented within the code Basilisk (Popinet, 2015) which provides a set of adaptive-multigrid solvers on quad/oct-trees. The method is validated against various test cases involving spherical particles in different flow regimes: from Stokes flow to highly inertial flows. To compute flows laden with non-spherical particles, we couple our AMR-DLM/FD solver to the granular solver Grains3D (Wachs et al. 2012). With this numerical tool, the dynamics of multiple particles of complex shape freely moving in a fluid on adaptive quad/oct-tree becomes accessible. As an illustration, we present the case of 600 free falling cubes in a large container.

12:02PM S28.00008 Chaotic Orbits of Tumbling Ellipsoids in Viscous and Inviscid Fluids\textsuperscript{1}, ERICH ESSMANN, PRASHANT VALLURI, School of Engineering, University of Edinburgh, STEPHANE POPINET, Sorbonne Université, Institut Jean le Rond d’Alembert, RAMA GOVINDARAJAN, ICTS-TIFR, Bangalore — It was shown that the equations of motions of an immersed tri-axial ellipsoid become non-integrable under certain inviscid conditions, (Kozlov and Onishchenko, 1982). Non-integrability is a strong function of density ratio and the initial energy ratio. In viscous systems, we have noted evidence of chaotic orbits for symmetric ellipsoids. Due to vortex shedding behaviour in this context breaking the symmetry of the system. We show how chaos can be exploited under viscous environments to promote mixing.

\textsuperscript{1}EC-RISE-ThermaSMART
Presented.

Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

Facility at the University of Michigan. Experiments evaluating the drag produced by the live biofilm were then compared to those of solid, three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

Three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

ELIZABETH CALLISON, JOEL HARTENBERGER, JAMES GOSE, University of Michigan, MARC PERLIN, Texas A&M University, STEVEN CECCIO, University of Michigan — Soft biofilms can form at flow boundaries, producing increased friction drag which can adversely affect the performance of hydrodynamic systems. The underlying mechanisms of drag production in soft biofilms are not well understood. Surface roughness, compliance, and the presence of streamers within the boundary layer flow can all contribute to the development of friction drag by turbulent boundary layers. To examine the drag producing flow, flat plates covered in biofilms were studied in the Skin-Friction Flow Facility at the University of Michigan. Experiments evaluating the drag produced by the live biofilm were then compared to those of solid, three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

Facility at the University of Michigan. Experiments evaluating the drag produced by the live biofilm were then compared to those of solid, three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

Facility at the University of Michigan. Experiments evaluating the drag produced by the live biofilm were then compared to those of solid, three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

Facility at the University of Michigan. Experiments evaluating the drag produced by the live biofilm were then compared to those of solid, three-dimensional printed, rigid replicas to differentiate the measured drag forces and their components. These rigid replicas were generated via additive manufacturing using in situ measurements of the biofilm surface profile collected at several flow speeds and growth incubation times. The hydrodynamic performance of the biofilm was determined through pressure drop measurements as well as planar particle image velocimetry of the channel flow. Comparisons of the resistance curves for the rigid replicas and live biofilm will be discussed and flow measurements will be presented.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.

This work was supported by NASA under cooperative agreement number NNX16AQ96A.
11:36 AM S29.00006 Ion-Mediated Swelling in a Model of Gastric Mucus Gel. OWEN LEWIS, Florida State University, JAMES KEENER, AARON FOGELSON, University of Utah — The gastric mucus layer serves a protective function in the human stomach, shielding the epithelium from the digestive machinery of the stomach, but it is not understood how this gel layer is maintained in vivo. Mucus degradation via digestion of the gel network at the lumen must be balanced by secretion and swelling of new mucus at the stomach wall. These processes are dependent on the local ionic composition of the gel solvent, which varies throughout the layer. Here, we present a comprehensive model of mucus-like polyelectrolyte gel based on a two-phase framework. This model extends classical theory to account for the dependence of the Flory interaction parameter and standard free energies on transient, ion-mediated crosslinks within the gel network. We present a computational investigation of the dynamic swelling behavior of the model. In particular, we quantify the rate at which a globule of crosslinked gel swells when exposed to an ionic bath as a function of bath concentration and network chemistry. We show that the swelling rate has a non-monotone dependence on the molarity of the bath solution, in part due to the existence of an additional chemical pressure not predicted by classical theory.

11:49 AM S29.00007 Active carpets, part 1: Transport driven by bacterial clusters and topological defects1. FRANCISCA GUZMAN-LASTRA, Universidad Mayor, ARNOLD MATHIJSSEN, Stanford University, ANDREAS KAISER, Retired, HARTMUT LOEWEN, Heinrich-Heine University — Biological activity is highly concentrated on surfaces, from molecular motors and ciliary arrays to sessile suspension feeders and biofilms; together they form the class of ‘active carpets’. While the physics of active suspensions has raised considerable interest, it remains unclear how energy and momentum injection from active surfaces can drive living systems out of equilibrium. Here we demonstrate that active carpets of bacteria or self-propelled colloids generate coherent flows towards the substrate, and we propose that these currents provide efficient pathways to replenish nutrients that feed back into activity. A full theory is developed in terms of gradients in the active matter density and velocity, and applied to bacterial turbulence, topological defects and clustering. Currents with complex spatiotemporal patterns are obtained, which are tunable through confinement. Our findings show that diversity in carpet architecture is essential to maintain biofunctionality.

1 A. M. acknowledges funding from the Human Frontier Science Program (Fellowship LT001670/2017). F. G. L. acknowledges Millennium Nucleus Physics of active matter of the Millennium Scientific Initiative of the Ministry of Economy, Development and Tourism, Chile. H. L. acknowledges support from the Deutsche Forschungsgemeinschaft, DFG Project No. LO 418/17-2

12:02 PM S29.00008 Active carpets, part 2: Generalisation of Fick’s laws for enhanced diffusion and particle capture by life at interfaces1, ARNOLD JTM MATHIJSSEN2, Stanford University, FRANCISCA GUZMAN-LASTRA, Universidad Mayor, Santiago, Chile, MANU PRAKASH, Stanford University, HARTMUT LOEWEN, Heinrich Heine University Düsseldorf — Fluctuations lie at the core of biological processes, facilitating diffusive transport and driving the cell’s molecular machinery. Fick’s laws of diffusion are well established in classical thermodynamics, but living systems operate far from thermal equilibrium as energy is injected locally by activity, which can give rise to surprising effects not captured by passive diffusion. Especially on surfaces this metabolic activity is highly concentrated, from molecular motors through ciliary arrays to sessile suspension feeders and bacterial biofilms, together forming the class of ‘active carpets’. Here, we consider the flows generated by these different active surfaces and show how they can enhance diffusion, directed transport, and particle capture. We derive the diffusion coefficients as a function of distance from these active carpets and we formulate the corresponding generalised Fick’s laws. These laws feature remarkable solutions, including non-Boltzmannian sedimentation profiles with particles hovering a finite distance above the active carpet, and enhanced particle capture by a raised diffusive flux. Our results shed new light on the non-equilibrium properties of materials with active boundary conditions and life at interfaces.

1 AJTMM acknowledges funding from the Human Frontier Science Program (Fellowship LT001670/2017).
2*AJTMM and FGL contributed equally to this work

Tuesday, November 26, 2019 10:31 AM - 12:02 PM
Session S31 Biological Fluid Dynamics: Collective Behavior 613 - Hamid Karani, Northwestern University

10:31 AM S31.00001 Shrinking spinning fire ant rafts. HUNGTANG KO, DAVID HU, Georgia Institute of Technology — Fire ants make rafts to stay afloat during flooding seasons. The ability to respond to different fluid environments is critical to raft sustainability. To investigate the response, we built an experiment setup to create two fluid conditions for the fire ant raft: rigid-body rotation and Taylor-Couette vortex. We found that following a rapid expansion phase, the fire ant raft shrinks at a much longer time scale under all conditions. We discovered that the additional shear from the Couette vortex help stabilize the ant raft while the centrifugal force didn’t have appreciable effects. Furthermore, the result suggests that rotation can inhibit the exploration behavior of individual fire ants on the raft.

10:44 AM S31.00002 Insights on rotor ensemble dynamics using a new scalable computational platform. WEN YAN, MICHAEL SHELLEY, Flatiron Institute, Simons Foundation — Suspensions of Stokes rotors consist of immersed particles that are driven to rotate, with that rotation creating flow fields that can create large-scale coupling and dynamics. Such rotor systems are typically driven by external means, such as a rotating magnetic field. Here we study the dynamics of closely packed rotor systems using a new method that combines a high-order accurate fluid solver, based on integral equation methods, and a temporally stable particle-particle collision solver based on geometric constraint optimization. This new computational technique is scalable on parallel computer clusters and allows us to simulate the development of large-scale dynamics. We first report the internal dynamics of a monolayer consisting of 10,000 rotors, each driven by a torque perpendicular to the monolayer. This shows both large-scale collective dynamics and complex small-scale interactions. In the second example, we turn the torque sideways and find a Kelvin-Helmholtz-like instability of the monolayer induced by the particles’ rotational flows and steric interactions.
On the locomotion and collective behavior of biopolymer-producing bacteria

JUANES, MIT — Viscous environments are ubiquitous in nature and engineering applications — such as mucous in lungs and oil recovery systems; ranging from swarms and jets to dynamic clusters and vortices. We elucidate the role of complex physical interactions between colloidal particles and show that different emergent states are identified by competing characteristic time and length scales. More specifically, we show that the time scales during the run and tumbling phases play a major role in establishing different stable collective states. Our findings show the potential for dynamic transitioning between states at constant concentration and activity (speed) of active particles by solely tuning the kinematic time and length scales of individual random walkers.

supporter by NSF awards CBET-1704996 and CMMI-174001

Rheology of Bacteria Superfluids in Viscous Environments

CHUI, MIT, KAREN FAHRNER, Harvard University, CARINE DOUARCHE, HAROLD AURADOU, Université Paris-Sud, CNRS, RUBEN JUANES, MIT — Viscous environments are ubiquitous in nature and engineering applications — such as mucous in lungs and oil recovery strategies in the earth’s subsurface — and in all these environments, bacteria also thrive. It has been well documented that active suspensions of bacteria can behave as a superfluid, in terms of reducing the viscosity of the surrounding fluid by their collective motion, but it is not known what their effect is when they are introduced to a viscous environment. Here, we investigate experimentally how viscous environments can change the ability of pusher-type bacteria (E. coli) in creating a superfluid regime. Using a Couette rheometer, we measure stress as a function of the applied shear rate, and define the apparent viscosity of $E. coli$ suspensions, varying both the density of the bacteria population within the suspension and the viscosity of the suspending fluid. We find that the bacteria suspensions remain capable of behaving as a superfluid by reducing their surrounding viscosity to zero, and that changes in solvent viscosity mainly affects the range of shear rates over which the superfluid regime is possible. From the data, we assemble the ingredients needed to build a theoretical model that describes the effective viscosity of an active fluid as a function of the bacteria density and its environment (shear rate, solvent viscosity). Beyond the value for developing a theoretical model, our results open the possibility of giving an empirical guidance for numerical simulations involving bacteria and fluid flow.

1This work is supported by the National Science Foundation Grant CMMI-1562871.

Predator-Prey Interactions using Deep Reinforcement Learning

LOO, MOHAMMAD-REZA ALAM, University of California, Berkeley — Here we unveil synergistic cooperation of micro-swimmers to form a stealth swarm that minimally disturbs the surrounding fluid. We call this mode of swarming the ‘concealed mode, which can be achieved when a group of swimmers actively collaborate to cancel out one another’s disturbing flows. We then demonstrate how such a concealed swarm can remain stealth while actively gathered around a favorite spot (e.g. a nutrient source), pointing toward a target (e.g. attacking a prey flock), or tracking a desired trajectory in space. Our findings also provide a clear road map to control and lead stealth flocks of swimming micro-robots formed through their active collaboration in minimally disturbing the host medium.

This work is supported by the National Science Foundation Grant CMMI-1562871.
10:31AM S32.00001 Space-time characteristics of wind turbulent pressure field at wave surface. XUANTING HAO, LIAN SHEN, University of Minnesota — The wind pressure field at the wave surface, with fluctuations several orders of magnitude smaller than the mean atmospheric pressure, has a notable impact on the wave energy growth. The time-variant and irregular wave field poses great challenge to the full space-time measurement of the surface pressure. We present a simulation-based study of the space-time characteristics of the surface pressure field. The numerical tool is a combined turbulence solver and wave-phase-resolved model, where the wind turbulence is simulated with large-eddy simulation and the nonlinear wave field simulated using a high-order spectral method. The joint probability distribution of the surface pressure and the vertical surface velocity is found to resemble that of turbulent flows over a compliant wall. We also conduct spectral analysis on the surface pressure field to obtain the wavenumber-frequency spectrum. While the turbulent pressure fluctuations have a convection velocity comparable to turbulent flows over a fixed flat wall, a distinct wave effect is identified. The wavenumber/frequency peak of the wave-coherent pressure field is found to deviate from that of the wave field, suggesting nonlinearity in the turbulence-wave interactions.

10:44AM S32.00002 Effect of the presence of long waves on the wave growth rate of short waves. TAO CAO, LIAN SHEN, University of Minnesota — We performed a DNS study of the turbulent wind over two slowly progressive water waves with different wave lengths. It is shown that the presence of the long wave results in a significant reduction of the form drag on the short wave. To explain the underlying physical mechanism, we obtained the momentum equations for the wave-coherent motions induced by the long wave and the short wave, respectively. We first showed that the asymmetric wave-induced pressure across the surface wave crest is responsible for the form drag on the wave surface and is mainly caused by the wave-induced advection associated with the vertical wave-induced velocity that is in-phase with the surface elevation. It is further found that in the presence of the long wave, the wave-induced advection corresponding to the short wave is significantly reduced, which results in a smaller asymmetric pressure on the short wave, and thereby a lower form drag. Moreover, we observed that the extent of the reduction of the form drag on the short wave depends on the wave age of the long wave. The smaller the wave age of the long wave, the larger the reduction of the form drag on the short wave.

10:57AM S32.00003 Numerical simulations of the turbulent flow over an ocean wavefield. DEVIN CONROY, LAURA BRANDT, JAMES ROTTMAN, Leidos — The air-sea dynamics for wind driven ocean waves are crucial for understanding and predicting the weather and global climate, as well as for predicting forces on ships and offshore wind farms for design and operation. The wind-wave problem has been an active area of research for many years, spanning observational, experimental, theoretical, and numerical studies. Here we leverage the Large Eddy Simulation technique with a sub-grid scale sea surface roughness model for the interfacial stress and an interface capturing Volume-of-Fluid method to obtain highly accurate phase resolved simulations over a sufficiently large domain size. We investigate the dynamics of the air-sea interaction, starting with a JONSWAP spectrum, to understand the generation of turbulence, wind-wave growth, non-linear surface dynamics, and wave breaking on the overall energy budget. We find that the wind stress tends to add energy at small length scales (<0.1 m), due to the surface shear and turbulent pressure fluctuations in the air above the surface. Preliminary results also indicate that under strong wind forcing (or low wave age), air separation occurs near wave crests that later undergo breakage.

1Funding from the Office of Naval Research

11:10AM S32.00004 ABSTRACT WITHDRAWN

11:23AM S32.00005 A kinetic approach to estimate air-sea exchanges driven by sea spray in high winds. FABRICE VERON, University of Delaware, LUC MIEUSSENS, University of Bordeaux, France — Sea-spray is known to be a fundamental component of air-sea heat flux in high wind speed conditions where water drops are frequently ejected from the sea surface because of breaking waves and breaking related phenomena such as bubble entrainment and whitecaps. Once ejected from the ocean surface, these drops are transported and dispersed in the Atmospheric Boundary Layer (ABL) where they interact and exchange momentum, heat, and moisture with the ambient atmosphere. However, understanding of these spray fluxes pathways, and our ability to model them remains limited. In this work, we borrow the framework from established kinetic gas theory, and apply these mathematical tools to model the transport of spray droplets and the exchanges of heat, momentum, and moisture between the drops and the atmosphere. Within this framework, one of the most vexing component of this spray flux problem, i.e. the feedback from the drops on the atmosphere, is relatively straightforward to account for. This work is largely exploratory and in the early stages of development. We will present an overview of the approach as well as preliminary results.

11:36AM S32.00006 An Experimental Study of Droplet Generation by Plunging Breaking Water Waves, MARTIN A. ERININ, SOPHIE WANG, REN LIU, DAVID TOWLE, XINAN LIU, JAMES H. DUNCAN, University of Maryland — The production of droplets by strong, moderate, and weak plunging breakers is studied experimentally in a laboratory. The water waves are generated mechanically using a dispersively focused wave packet technique with an average wave packet frequency of 1.15 Hz for all three waves. Surface profile histories of the breaking wave crests are measured using a cinematic laser-induced fluorescence technique. The temporal evolution of the phase averaged surface profile of the breaker, obtained from 10 runs for each wave, are used to characterize the breakers. Droplets are measured using a cinematic digital in-line holographic system positioned at 26 streamwise locations along a horizontal plane (herein called the measurement plane) that is 1 cm above the maximum wave crest height. The droplet radii (r > 100 μm), positions and trajectories are determined from the holograms. Counting only the droplets that are moving up across the measurement plane, the spatio-temporal distribution of droplet generation by the breakers is obtained. Droplet statistics including total number, mean diameter and speed are presented. The relative importance of the various droplet generation mechanisms in the three waves are discussed and correlated with the mean wave profile characteristics.

1The support of the Division of Ocean Sciences of the National Science Foundation under grants OCE0751853 and OCE1925060 is gratefully acknowledged.
The amplitudes of individual modes are defined such that the total standard deviation of the two perturbations are the same and all modes are initially linear. The DNS results are compared with implicit large eddy simulations (ILES) of the same initial conditions in the high Reynolds number limit case for comparisons is shock interaction with a solid cylinder. Langmuir circulation (LC) beneath an initially quiescent air-water interface appear shortly after the initiation of wind-driven gravity-capillary waves and provide the laminar-turbulent transition in wind speeds between 3 and 6 m/s. LES reveals Langmuir turbulence characterized by multiple scales ranging from small bursting eddies at the surface that coalesce to give rise to larger (centimeter-scale) LC over time. The growing LC scales lead to increased vertical scalar transport at depths below the interface and thus greater scalar transfer efficiency. Simulations were performed with a fixed wind stress corresponding to a 5 m/s wind speed but with different wave forcing parameters. It is observed that longer wavelengths lead to more coherent, larger centimeter-scale LC providing greater contribution to the turbulent vertical scalar flux. In all cases, the molecular diffusive scalar flux at the water surface relaxes to the same statistically steady value after transition to Langmuir turbulence occurs, despite the different wave forcing across the simulations.

12:02PM S32.00008 Large-eddy simulation (LES) of small-scale Langmuir circulation and scalar transport1, ANDRES TEJADA-MARTINEZ, AMINE HAFSI, University of South Florida, CIGDEM AKAN, University of North Florida, FABRICE VERON, University of Delaware — Previous laboratory experiments have revealed that small-scale (centimeter-scale) Langmuir circulation (LC) beneath an initially quiescent air-water interface appear shortly after the initiation of wind-driven gravity-capillary waves and provide the laminar-turbulent transition in wind speeds between 3 and 6 m/s. LES reveals Langmuir turbulence characterized by multiple scales ranging from small bursting eddies at the surface that coalesce to give rise to larger (centimeter-scale) LC over time. The growing LC scales lead to increased vertical scalar transport at depths below the interface and thus greater scalar transfer efficiency. Simulations were performed with a fixed wind stress corresponding to a 5 m/s wind speed but with different wave forcing parameters. It is observed that longer wavelengths lead to more coherent, larger centimeter-scale LC providing greater contribution to the turbulent vertical scalar flux. In all cases, the molecular diffusive scalar flux at the water surface relaxes to the same statistically steady value after transition to Langmuir turbulence occurs, despite the different wave forcing across the simulations.

1This research was made possible by the NSF.

Tuesday, November 26, 2019 10:31AM - 12:15PM
Session S33 Flow Instability: Richtmyer-Meshkov, Computational

10:31AM S33.00001 Direct numerical simulation of Richtmyer-Meshkov instability with broadband initial perturbations, MICHAEL GROOM, BEN THORNBERRY, The University of Sydney — The effects of Reynolds number on the early to intermediate time behaviour of a mixing layer induced by Richtmyer-Meshkov instability (RMI) are investigated through a series of direct numerical simulations (DNS) using the finite-volume code FLAMENCO. This study presents, for the first time, results from direct numerical simulations of RMI evolving from amplitude perturbations containing a broad bandwidth of initial modes. In particular, two different broadband perturbations are analysed, defined by their initial radial power spectra $P(k) = C_k^m$ where $m = -1, -2$. The amplitudes of individual modes are defined such that the total standard deviation of the two perturbations are the same and all modes are initially linear. The DNS results are compared with implicit large eddy simulations (ILES) of the same initial conditions in the high Reynolds number limit, showing that although there is significant suppression of turbulence in the low Reynolds number DNS, the growth in integral component for further insight into droplet dynamics and exchange fluxes.

1Support from US National Science Foundation grants 1235039 and 1233808 is gratefully acknowledged.

10:44AM S33.00002 Additive Manufacturing and Richtmyer-Meshkov Initial Condition Studies, TIFFANY DESJARDINS, ADAM MARTINEZ, JOHN CHARONKO, KATHY PRESTRIDGE, Los Alamos National Laboratory — With the advances in materials and additive manufacturing we have begun to re-explore the application of membranes in RMI experiments. At the Vertical Shock Tube facility, we are using AM techniques to develop shaped membranes for studies of different types of initial interfaces, motivated by the need to explore non-diffuse initial conditions with specific geometrical configurations. The need for understanding of features in applications such as inertial confinement fusion (ICF) drives this work. The goal is the development of a barrier at the initial interface that repeatedly breaks, minimally influences the growth of the instability, and can be shaped to look at desired interface configurations. We have found a material fragile enough that when hit with an M = 1.2 shock, it is returned to a near complete dust state. The particles are heavy enough to lag behind the interface region and preliminary experiments studying single modes will be presented.

10:57AM S33.00003 Validation of a Simplified WENO Scheme with Artificial Viscosity for a Shock Interface Interactions1, BRIAN ROMERO, SVETLANA POROSEVA, PETER VOROBIEFF, University of New Mexico — The goal of this work is to validate numerical simulation of the shock-driven evolution of an initially axisymmetric cylindrical particle cloud (or a heavy gas column) against laboratory experiments. In the initial conditions in the experiment, there is a density gradient in gas, an average density gradient due to the particle seeding, or both. The cylindrical column is comprised of a mixture of sulfur hexafluoride, acetonitrile, and air, and surrounded by air, leading to the representative Atwood number of 0.61. The Mach number in experiments varied from 1.2 to 2.0. Simulations were conducted at Mach numbers matching the experiment using a simplified WENO scheme incorporating a new C-method for artificial viscosity. The initial modeling/validation exercises featured a two-dimensional simulation of shock interaction with the gas column, with variations in the level of initial diffusion on the density interface. From a sharp interface to a diffuse boundary matching the conditions measured in experiment. Further studies include 3D problems with multiple gas species and immersed boundary conditions, and shock wave refraction for various values of the gas constant $\gamma$. The limit case for comparisons is shock interaction with a solid cylinder.

1This work is supported by the Defense Threat Reduction Agency (DTRA) grant HDTRA1-18-1-002.
11:10AM S33.00004 Numerical Simulation of Richtmyer-Meshkov Instability in Converging Geometries, ZUOLI XIAO, JINXIN WU, College of Engineering, Peking University — A high-order turbulence solver in curvilinear coordinates is developed for numerical simulation of multi-species compressible flows with discontinuities, in which high-order compact finite difference schemes and localized artificial diffusivities are employed to satisfy the need for high accuracy and discontinuity capturing. The Richtmyer-Meshkov instability (RMI) induced mixing flows driven by implooding shock in both cylindrical and spherical geometries are numerically investigated using direct numerical simulation method. The detailed evolutions of RMI are compared with experimental data and theoretical results, and reasonably good consistency is observed both qualitatively and quantitatively before reshock. The vortex dynamics of RMI after shock impingement is also discussed, and an initially limited circulation on the interface during shock-interface interaction is calculated from the simulation, which is in accordance with the results of theoretical prediction. Moreover, effects of the modes and amplitude of initial perturbations, as well as the incident shock Mach number on the interfacial growth rate are evaluated during the shock implosion and reflection from the center.

11:23AM S33.00005 A Comparison of Two- and Three-Dimensional Single-Mode Reshocked Richtmyer-Meshkov Instability Growth, OLEG SCHILLING, Lawrence Livermore National Laboratory — The growth dynamics of two- and three-dimensional single-mode reshocked Richtmyer-Meshkov instability are compared using data from high-resolution implicit LES of a model of the Mach 1.3 air(acetone) and sulfur hexafluoride Jacobs and Krivets shock tube experiment. The numerical amplitude growth is compared to the predictions of several nonlinear instability growth models. The dynamics of reshock are described, and the post-reshock mixing layer amplitude growth rate is compared to the predictions of several reshock models. It is shown that using two-dimensional simulations to understand three-dimensional dynamics is valid only at early-to-intermediate times before reshock. At intermediate-to-late times the three-dimensional growth is larger than the two-dimensional growth. The reshock dynamics are also different between two and three dimensions.

11:36AM S33.00006 Effects of the secondary baroclinic vorticity on turbulent energy cascade in the Richtmyer-Meshkov instability, NAIFU PENG, YUE YANG, Peking University — We study turbulent mixing in the Richtmyer-Meshkov instability (RMI) induced by a planar shock wave at Mach 1.5 with multimode interfacial perturbations between air and SF6. By separating different types of the perturbations, we develop a double-density model for simplifying the RMI, and find that the effects of the secondary baroclinic vorticity (SBV) play an important role during the flow evolution. The SBV, caused by the misalignment of the pressure gradient produced by the velocity perturbation and the density gradient near the interface, leads to the nonlinear evolution of the interface with the generation of spike- and bubble-like structures. Moreover, the SBV produces small-scale vortical structures and affects the turbulent energy cascade in the mixing zone. The kinetic energy spectrum affected by the SBV is closer to the —μ/3 scaling law than the classical —5/3 law for constant-density homogeneous isotropic turbulence.

11:49AM S33.00007 Computational study of the interactions of two bubbles along an interface undergoing the Richtmyer-Meshkov instability, MICHAEL WADAS, ERIC JOHNSEN, University of Michigan — In addition to its prevalence in astrophysics, the shock-driven growth of interfacial perturbations (the Richtmyer-Meshkov instability, or RMI) has important practical consequences in applications of fusion research. Our objective is to numerically investigate the generation of vortex dipoles that escape the confines of the mixing region, as well as the reacceleration of bubbles, by focusing on the interaction of two adjacent bubbles of different sizes subjected to the RMI. Our hypothesis is that the escape of a vortex dipole can be predicted from the initial conditions based on the vorticity associated with the bubbles. Simulations are performed using an in-house, high-order accurate Discontinuous-Galerkin code. We demonstrate deviations from existing bubble merging models in the non-linear regime caused by the interaction of adjacent pairs of spikes, which had not previously been considered. We further develop a criterion that predicts the regime that will emerge for a given interface.

12:02PM S33.00008 Impulse-driven Richtmyer-Meshkov instability in Hall-magnetohydrodynamics, NAIJIAN SHEN, DALE PULLIN, California Institute of Technology, VINCENT WHEATLEY, University of Queensland, RAVI SAMTANEY, King Abdullah University of Science and Technology — We utilize the incompressible, Hall-MHD model for conducting fluids to investigate the effect of Hall current on the stability of an impulsively-accelerated, perturbed density interface, or contact discontinuity (CD) separating two fluids in the presence of a background magnetic field. This is used as a simple model, in a conducting fluid, of a Richtmyer-Meshkov (RM) flow that is characterized in a neutral-fluid by a shock-wave-density-interface interaction. The linearized equations of motion are formulated for a sinusoidal interface perturbation, and then solved as an initial-value problem using a Laplace transform method. The presence of the magnetic field is found to suppress the incipient interfacial growth associated with neutral-gas, RM instability (RMI). When the ion skin depth is finite, the vorticity dynamics that drive the suppression of the RMI differs markedly from the ideal MHD, RM flow. Hall MHD allows the presence of a tangential slip velocity leading to finite circulation deposition at the CD. Vorticity is produced by the perturbed magnetic fields and transported to infinity by a dispersive wave system leading to decay of the velocity slip at the interface with the effect that that interface growth remains bounded but distorted by damped oscillations.

KAUST Office of Sponsored Research
Numerical simulations are carried out, which reveal the development of large-amplitude waves in the axisymmetric, and non-axisymmetric cases. Extensions to non-isothermal situations are outlined.

1Royal Academy of Engineering, UK, Research Chair in Multiphase Fluid Dynamics for OKM

10:44AM S34.00002 Reduced-order Models for Two-phase Annular Flows in Vertical Pipes 1

1. THOMAS EWERS, Imperial College London, ALEXANDER WRAY, University of Strathclyde, OMAR MATAR, Imperial College London — We study the dynamics of two-phase annular flow in vertical pipes. The conditions considered are such that there is no mass exchange between the phases due, for instance, to liquid entrainment into the gas, and bubble entrainment within the liquid. The gas is assumed to be turbulent, whilst the liquid phase exists in the form of a thin film adjacent to the wall. Reduced-order models are derived using asymptotic reduction for both axisymmetric and non-axisymmetric cases. The turbulence in the gas is modelled using a mixing length relation, while the method of weighted residuals is used in the film wherein inertial contributions are significant but the flow remains laminar. Numerical computations are carried out, which reveal the development of large-amplitude waves in the axisymmetric, and non-axisymmetric cases. Extensions to non-isothermal situations are outlined.

10:57AM S34.00003 Analysis of Blast Driven Instability and Mixing from the Energetic Dispersal of a Perturbed Particle Bed

1. FREDERICK OUELLET, RAHUL BABU KONERU, JOSHUA GARRNO, S. BALACHANDAR, University of Florida, BERTRAND ROLLIN, Embry-Riddle Aeronautical University — The evolution of particle clouds following interactions with a blast wave and contact interface resulting from detonating a high-energy explosive is a difficult problem for both numerical simulations and physical experiments. Experimentally, it is challenging to accurately characterize the initial states of both the explosive and the surrounding particle bed. Limitations also exist in the available diagnostic tools and measurable data which can be extracted from experiments. Thus, simulations can be a cheaper method to analyze the physics governing the interactions between the expanding particle cloud and the highly compressible, post-detonation fluid flow. Using multiphase, compressible flow simulations in an Eulerian-Lagrangian frame, the impact of perturbing a particle bed surrounding an explosive charge is analyzed. The analysis focuses on the multiphase instabilities and late-time behavior displayed by the dispersing particle cloud and discusses the associated underlying physical phenomena. Effects of the instabilities on the mixing behavior of the detonation products with the surrounding air are also discussed. The perturbations are varied to unravel the effects of the initial particle distribution and its persistence in the late time particle cloud and the background fluid flow. Inspired by work on two-fluid interfacial instabilities, this study relates to work in the emerging field of shock-driven multiphase instabilities but at extreme conditions and moderate initial particle loadings.

11:00AM S34.00004 On hydrodynamic instabilities in pseudo-boiling with supercritical fluids 1

1. REBECCA BARNEY, UC Davis, ROBERT NOURGALIEV, Lawrence Livermore National Laboratory, JEAN-PIERRE DELPLANQUE, UC Davis, ROSE MCCALLEN, Lawrence Livermore National Laboratory — We investigate hydrodynamic instabilities arising in mixed forced and natural convection laminar flow at supercritical thermodynamic conditions. Fluid in this regime is compressible, with highly varying properties, even for vanishingly small Mach numbers. Hydrodynamic stability is influenced by the occurrence of pseudo-phase change near the channel wall, as defined by the peak of the specific heat above the critical point. While the fluid properties do not explicitly exhibit a discontinuous change, the steep continuous property changes create flow patterns which qualitatively look like boiling. Of interest is investigating this boiling-like phenomenon to characterize the heat transfer in the supercritical regime. Due to the highly-varying density and specific heat fields, we solve the fully compressible Navier-Stokes equations. An advanced equation of state for supercritical water was implemented in an Arbitrary Lagrangian-Eulerian multi-physics simulation tool developed at Lawrence Livermore National Laboratory. A newly developed, robust, 5th order in both space and time, fully implicit, all-speed, reconstructed discontinuous Galerkin method is used to enable the numerical simulations. As the bottom wall is heated, the density decreases at the wall increasing the flow instabilities. The Richardson number indicates when the flow is dominated by forced or natural convection and provides a map/correlation between buoyancy effects and unstable flow features.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344. Information management release number LLNL-ABS-78338.

11:23AM S34.00005 A new mechanism for the generation of interface distortions in liquid jets 1

1. HANUL HWANG, PARVIZ MOIN, M. J. PHILLIPP HACK, Center for Turbulence Research, Stanford University — The onset of the atomization process of liquid jets is commonly understood as a sequence of exponential instabilities whose amplification eventually leads to a distortion of the interface and the breakup of the jet into droplets. Our study analyzes the amplification of interface perturbations of liquid jets through an optimization problem within a spatial linear framework. The objective functional is the interface potential energy due to surface tension. We demonstrate that a multi-phase Orr mechanism can serve as an alternative pathway for the generation of interface distortions by redistributing energy from the mean shear into perturbations of the jet surface. Parameter studies show that the amplification of interface disturbances scales with both Reynolds number and Weber number. Analysis of the budget of the perturbation kinetic energy provides further insight into the underlying physics. For high Weber numbers, the amplification of the surface distortion is bounded by viscous effects, whereas surface tension limits the growth in the case of low Weber numbers.

1This research is supported by the Franklin P. and Caroline M. Johnson Fellowship.
11:36AM S34.00006 Squeezing to Bending Transitions of Interfacial Electrohydrodynamic Instabilities for Digitization and Mixing of Two-Phase Microflows1. JOYDIP CHAUDHURI2, TAPAS KUMAR MANDAL2, DIPANKAR BANDYOPADHYAY2, IIT Guwahati. — External field induced interfacial instabilities have shown significant potential in the miniaturization of flow patterns inside the microfluidic devices. Electric field induced instabilities in a trilayer oil-water microflow is explored with the help of analytical models and computational fluid dynamics simulations. Twin oil-water interfaces undergo either in-phase bending or anti-phase squeezing mode of deformation when a direct current (DC) electric field is applied locally. The selection of modes depends on the magnitudes of applied DC field intensity and oil-water interfacial tension. The growth of the squeezing mode leads to a time-periodic dripping of droplets at lower field intensities, whereas, bending mode develops into ‘whiplash’ ejection of miniaturized droplets having octuplet microvortices inside and outside, at higher field intensities. A transition from purely laminar flow is observed during the switch over to bending mode, resembling von Kármán vortex street formation. Use of alternating current (AC) electric field with variation in frequency and waveform is also found to create on-demand and time-periodic array of flow features following the mode selection.

1DST SERB Grant no. EMB/2016/001824, Government of India
2Chemical Engineering Department, IIT Guwahati, India

11:49AM S34.00007 Phase transitions to condensate formation in two-dimensional turbulence1. MORITZ LINKMANN, Fachbereich Physik, Philipps-University of Marburg, GUIDO BOFFETTA, Dipt. Fisica, University of Torino, Italy, M. CRISTINA MARCHETTI, Dept. Physics, UC Santa Barbara, USA, BRUNO ECKHARDT, Fachbereich Physik, Philipps-University of Marburg — Two-dimensional (2d) and quasi-2d flows occur at macro- and mesoscale in a variety of physical systems. Examples include stratified layers in Earth’s atmosphere and the ocean, soap films and more recently also dense bacterial suspensions, where the collective motion of microswimmers induces patterns of mesoscale vortices. A characteristic feature of 2d turbulence is the occurrence of an inverse energy cascade. In absence of large-scale friction the inverse energy cascade results in the formation of large-scale coherent structures, so-called condensates. We here study the formation of the condensate as a function of the kind and amplitude of the forcing. Direct numerical simulations show that the condensate appears in a phase transition. For prescribed energy dissipation the transition is second order; for active matter, where the forcing is due to a small-scale instability, the transition is first order. The phase transition separates two markedly different types of 2d dynamics: in turbulence with a condensate, energy input is mostly balanced by dissipation in the condensate and intermediate scales follow an inertial cascade; without a condensate dissipation is spread over the intermediate scales and the properties of the energy transfer are different and non-universal.

1Supported by DOE FES grant number DE-SC0014318

Tuesday, November 26, 2019 10:31AM - 12:15PM
Session S35 Microscale Flows: Devices

10:31AM S35.00001 Hydrostatic and hydrodynamic bulge testing of pre-stressed thick elastic plates for microfluidic applications1. VISHAL ANAND, IVAN C. CHRISTOV, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana 47907, USA — Bulge tests are used to characterize material properties of elastic sheets. We present a theory of hydrodynamic bulge tests based on the fluid–structure interaction between a deformable top wall of a rectangular microchannel and a viscous Newtonian fluid flow within. Taking into account uniform isotropic pre-stress within the wall, we derive a model for the deformation using first-order shear-deformation theory for thick plates. It is shown that pre-stress reduces the transverse deformation. Then, the mechanics problem is coupled to flow under the lubrication approximation, in a perturbative manner, to capture both the effect of (non-constant) hydrodynamic pressure on the deformation and the effect of wall deformation on the channel’s hydrodynamic resistance. We obtain an analytical expression for the flow rate–pressure drop relation and show how it can be used to characterize the material properties (e.g., Young’s modulus or pre-tension) of an elastic sheet without measuring its deformation, which is difficult to do accurately at the microscale. Direct numerical simulations are performed using the commercial software ANSYS to explore the range of validity of the proposed bulge testing theory.

1This work was supported by NSF grant CBET-1705637.

10:44AM S35.00002 Theory of the bulging effect of soft microchannels with thick walls1. XIAOJIA WANG, IVAN C. CHRISTOV, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana 47907, USA — Long and shallow microchannels embedded in thick soft materials have been widely used in microfluidic devices. However, the bulging effect due to the fluid–structure interactions between the internal viscous flow and the soft walls has not been thoroughly understood. Previous models either contained a fitting parameter or were specialized to channels with thin walls. We present a theoretical study of the steady-state response of a deformable microchannel with a thick wall. Using lubrication theory for low-Reynolds-number flows and the linear elastic theory for isotropic solids, we obtain perturbative solutions for the flow and deformation. Specifically, only the channel’s top wall deformation is considered, and its thickness-to-width ratio is assumed to be ($t/w)^2 \gg 1$. Then, we show that the deformation at each stream-wise cross-section can be considered independently, and that the top wall can be regarded as a simply supported rectangle subject to uniform transverse pressure at its bottom. The stress and displacement fields are found using Fourier series. Then, the channel shape and the hydrodynamic resistance are calculated, yielding a new flow rate–pressure drop relation without any fitting parameters. Our results agree favorably with previously reported experiments.

1This work was supported by NSF grant CBET-1705637.
and provide examples of connection patterns in serial and parallel.

flow field. The design, based on a valve system, provides greater versatility and flexibility than previous designs. We modulized all the controllers and provide examples of connection patterns in serial and parallel.

11:10AM S35.00004 Flow manipulation via the effects of shape geometry and topology

1. TEJASWIN PARTHASARATHY, YASHRAJ BHOSALE, FAN KIAT CHAN, MATTIA GAZZOLA, University of Illinois, Urbana-Champaign — We demonstrate the importance of body curvature effects in achieving controlled flow manipulation in streaming settings. Here, we focus on microfluidic applications for inertial particle transport and mixing. We then challenge our understanding to explain prior experimental observations in two-dimensional flows. We further extend our exploration to three-dimensional flows where strategies related to body topology become accessible.

1 NSF CAREER Grant No. CBET-1846752 (MG)

11:23AM S35.00005 Regularized Inversed Holographic Volume Reconstruction for 3D flow diagnostics

1. JIARONG HONG, MALLERY KEVIN, University of Minnesota — We introduce regularized inverse holographic volume reconstruction (RIHVR) for 3D flow diagnostics using a single camera digital inline holographic particle tracking velocimetry (DIH-PTV). RIHVR solves an inverse problem whereby the 3D optical field best matching the recorded hologram is iteratively reconstructed utilizing the sparsity and spatial smoothness of the volume to regularize the solution. The reconstruction in RIHVR is substantially noise-free with improved axial resolution and increased maximum tracer concentration relative to prior DIH-PTV methods. The use of sparsity regularization in RIHVR enables a sparse data representation which reduces memory requirements and allows processing very large holographic images while simplifying the identification and tracking of individual particles. Using synthetic data, RIHVR shows a 40% improvement in localization accuracy and a 4x reduction in the RMS velocity fluctuation in addition to a threefold increase in the allowable tracer concentration. RIHVR further demonstrates its capability through measurements of microfiber dynamics with nanometer resolution in all three directions, swimming tracks of algae in a dense suspension, and the rotation rate of non-spherical particles in a T-junction flow.

1 Sponsored by National Science Foundation and University of Minnesota

11:36AM S35.00006 Understanding the Flow Mechanism in Micro Pulsating Heat Pipes Using Image Recognition

1. CHIHIRO KAMIJIMA, YUTA YOSHIMOTO, SHU TAKAGI, IKUYA KINEFUCHI, Department of Mechanical Engineering, The University of Tokyo — Heat generation in electronic devices has recently become a significant issue owing to their miniaturization and integration. Pulsating heat pipes (PHPs), which facilitate heat transfer via self-oscillation of liquid slugs, are promising devices thanks to their simple walkless structure and ease of miniaturization. However, complex and chaotic flow mechanism of PHPs has yet to be understood sufficiently, necessitating further investigation for device optimization. In this study, we fabricate a micro-PHP with a hydraulic diameter of 350 µm, and measure the PHP thermal conductivities using FC-72 as a working fluid under various conditions. We also visualize inner flows with a high speed camera and extract flow patterns using an image recognition technique. The results show that long and thin liquid films generated on the channel walls are of importance for effective heat transfer. Additionally, we model inner flows and heat transfer of the micro-PHP, and conduct numerical analyses using the extracted flow patterns. We find that latent heat transfer via the liquid films accounts for a significant portion of the overall heat transfer, while sensible heat transfer by the liquid slugs is negligible small.

11:49AM S35.00007 Experimental and numerical analysis of the reaction yield in a T-microreactor

1. ALESSANDRO MARIOTTI, CHIARA GALLETTI, ELISABETTA BRUNAZZI, ROBERTO MAURI, MARIA VITTORIA SALVETTI, DICI - University of Pisa — Microfluidic devices are attracting considerable interest in pharmaceutical and fine chemistry industry, because they allow continuous reactions with an unprecedented control over operating conditions. One critical issue in the use of microreactors is to obtain an efficient mixing and a high reaction yield while the flow being laminar, due to the small dimensions. In this context, the simplest and most studied configuration is the T-shaped one. The flow regimes occurring in these devices as the Reynolds number varies and the related mixing are well characterized in the literature. On the other hand, there is a little understanding of the effect of flow regimes on the yield of a chemical reaction. The present work is aimed at analyzing this aspect through the synergic use of experiments and numerical simulations. PIV measurements are used to characterize the flow pattern and optical techniques are employed to measure the reaction yield because the test-reaction is accomodated by discoloration. Simulations are based on finite-volume technique and local grid-adaptation. The chosen test-reaction is catalyzed by an acid in a homogeneous phase and this allow to easily investigate the combined effect of the Damkholer and of the Reynolds numbers on the reaction yield.

12:02PM S35.00008 A PDMS based passive microfluidic device for generating Platelet rich plasma

1. VIJAI LAXMI, SIDDHARTH TRIPATHI, SUHAS S JOSHI, AMIT AGRAWAL, Indian Institute of Technology Bombay, Bombay, Mumbai — Platelets are a pool of various growth factors which act as a soft tissue healer, and are also linked to various pathophysiological diseases. Platelet rich plasma (PRP) finds wide applications in platelet transfusion and biomedical research. Therefore, there is merit in selectively extracting platelets from blood. Clinically, platelets rich plasma is separated based on centrifugation process, which is however time consuming and leads to activation of platelets. The existing microdevices suffer from low throughput and have been tested only with dilute blood. Here, we design and test a passive PDMS based microdevice for obtaining PRP from whole human blood. The features involved in the microdevice are simple, and its fabrication involves a single layer of photolithography. The microdevice utilizes hydrodynamic forces for its operation, which are based on the biophysical laws and geometrical effects. The microdevice separates plasma with 14 - fold enrichment of platelets in it, while working in 0.2 ml/min to 0.4 ml/min flow rate range. The quality of the outlet sample is also checked by measuring the activation level of platelets, less than 5% platelets are found to be activated. These features make the presented microdevice unique.

1 Indian Institute of technology Bombay, Mumbai
than that of water-oil, allows for facile generation of microfibers. Needed for fibers based on water-oil systems. Moreover, the ultra-low interfacial tension of the ATPS, which is three orders of magnitude lower, enables many different microstructures. The all-aqueous nature of this system allows for generation of biocompatible materials, not requiring washing steps when preparing the fibers. The nonequilibrium all-aqueous fiber formation system may have important biotechnological applications, specifically for generation of scaffolds with well-defined microstructures of PEGDA fibers upon photopolymerization of the jet. We study the dynamics of the equilibration process, and the transient dynamics of these evolving fluid structures can be used as liquid templates to tune the size or the internal and surface properties of the fibers. A jet of a PEGDA-rich phase is formed in a continuous DEX-rich phase, inside a microfluidic device. As the ATPS equilibrates, a jet is formed due to the difference in interfacial tension. The jet is then jetted out of the device, and the microfibers are collected outside. The microfibers are then characterized using optical microscopy and scanning electron microscopy (SEM). This work confirms our previous work and shows that the nonequilibrium all-aqueous fiber formation system is a promising method for the fabrication of microfibers. This work presents a model for the nonequilibrium all-aqueous fiber formation system, including the transient dynamics of viscous peeling based soft actuators. The experimental data is compared with the suggested model, showing very good agreement.
of the field applied. In our work, we studied the influence of the field rotational frequency on the length and rotational speed of the chains as is a key factor in these dynamics. When it is increased, we observe the formation of rotating clusters with frequencies that differ from the one long chains aligned with the field. When the magnetic field is rotating, a competition between magnetic and viscous forces takes place, leading to the rotation of the chains following the magnetic field along with the reduction of their length. The frequency of the rotating magnetic field is a key factor in these dynamics. When it is increased, we observe the formation of rotating clusters with frequencies that differ from the one of the field applied. In our work, we studied the influence of the field rotational frequency on the length and rotational speed of the chains as well as on the main features of the clusters. In order to have the needed temporal resolution, we used a microscope combined with a high-speed camera. Thanks to this technology, we were able to determine a critical frequency between the two regimes mentioned above.

Superparamagnetic colloids under rotating magnetic field, FLORENCE MIGNOLET, GEOFFROY LUMAY, University of Liege — Colloids made of superparamagnetic micrograins offer the possibility to be controlled remotely with a magnetic field. This remote control enables various applications such as mixing, transport of fluids through microscale channels or propulsion in a viscous fluid. It is well known that when a constant magnetic field is applied on superparamagnetic colloids, they form long chains aligned with the field. When the magnetic field is rotating, a competition between magnetic and viscous forces takes place, leading to the rotation of the chains following the magnetic field along with the reduction of their length. The frequency of the rotating magnetic field is a key factor in these dynamics. When it is increased, we observe the formation of rotating clusters with frequencies that differ from the one of the field applied. In our work, we studied the influence of the field rotational frequency on the length and rotational speed of the chains as well as on the main features of the clusters. In order to have the needed temporal resolution, we used a microscope combined with a high-speed camera. Thanks to this technology, we were able to determine a critical frequency between the two regimes mentioned above.

Lateral migration of an electrophoretic particle in Newtonian and viscoelastic pressure driven flows, AKASH CHOUDHARY, Indian Institute of Technology Madras, DI LI, Clemson University, South Carolina, RENGANATHAN T, Indian Institute of Technology Madras, XIANGCHUN XUAN, Clemson University, South Carolina, PUSHPAVANAM S, Indian Institute of Technology Madras — In recent years there has been a growing interest towards incorporation of electrokinetics to inertial and viscoelastic focusing techniques to achieve higher throughput and external control over the focusing positions. This is performed by applying a DC electric field parallel to the flow. In this work, we investigate the effects of electrokinetics on the particle migration in pressure-driven flows of Newtonian and viscoelastic fluids through experiments and theoretical analysis. Using Lorentz reciprocal theorem in conjunction with perturbation expansion (in Reynolds or Deborah number), we derive analytical expressions which describe the lateral migration at the leading order. We find that that the direction of migration in inertial-Newtonian flow and inertialess-viscoelastic flow is towards the regions of high shear, provided the electrokinetic motion is in the direction of the imposed Poiseuille flow. The trajectories obtained from the theoretical analysis, in agreement with experiments, demonstrate that the interaction of electrokinetic and rheological effects can result in an enhancement in migration by an order of magnitude. Furthermore, it is revealed that the magnitude of the background shear and particle zeta potential primarily governs the migration.

**Tuesday, November 26, 2019 10:31AM - 12:15PM — Session S37 Particle Laden Flows: Simulations**

Particle number decomposition DEM parallel algorithm for particle laden flows, ZHENPING ZHU, University of Minnesota, Twin Cities; Xidian University, XIAOJING ZHENG, Xidian University, LIAN SHEN, University of Minnesota, Twin Cities — Domain decomposition is the most commonly used parallel algorithm in simulations of particle laden flows. However, there are two limitations of this algorithm. Firstly, particles are generally distributed unevenly over the different sub-domains in anisotropic flows, leading to problems of loading balance, especially when gravity effect is strong. Secondly, as particles naturally cross borders between sub-domains, passing values between sub-domains can be complicated. In this study, we develop a new particle number decomposition algorithm to overcome these two limitations. In this method, each core deals with almost the same number of particles to achieve efficient loading balance. We further develop a low storage method that can effectively store the particles information in each core, while substantially reducing the complexity of passing values among different cores. The new algorithm is easy to implement, and existing serial DEM algorithms remain unchanged in the parallel version. It also has high parallel efficiency and can compute millions of fully resolved particles.

Optimization of Finite Element Simulations of Colliding, Particle-laden Flows on Unstructured Grids, GRANT RYDQUIST, MAHDI ESMAILY, Cornell University — When simulating fluid systems in complex geometries, it is often necessary to discretize the system using a large number of unstructured elements to achieve accurate results. However, this can pose a number of problems when the simulation must also include Lagrangian particles. In such simulations, it is necessary to search for the element that contains a particle at each time step in order to accurately interpolate fluid properties to the location of the particle. While this operation is expensive for structured grids, in unstructured grids a standard implementation of this search algorithm requires $O(N_pN_e)$ operations, and can become prohibitively expensive for fine grids with many particles. A second situation which can lead to increased computational cost in simulations with Lagrangian particles is when binary collisions between particles must be captured. In general, each particle must be checked against every other particle to check for potential collisions, resulting in $O(N_p^2)$ calculations. In this work, we present a new technique to drastically reduce the computational cost in both of these situations through an optimization problem. In both of these cases, application of this optimization problem has led to a cost that is $O(N_p)$. 

**Tuesday, November 26, 2019 12:02PM - 1:00PM — Session S36 Microfluidics**
10:57AM S37.00003 Numerical modeling of inertial particles in under-expanded jets

YUAN YAO, University of Michigan, Ann Arbor, JASON RABINOVITCH, Jet Propulsion Laboratory, JESSE CAPECÉLÁTRO, University of Michigan, Ann Arbor — Inertial particle dynamics in flows that exhibit strong gas-phase compressibility and turbulence is crucial to many practical engineering applications. Such examples include coal dust explosions, shock wave lithotripsy, combustion/detonation, etc. Compared to their low-speed counterparts, particle-laden compressible flows like the examples listed here typically introduce new length- and time-scales and additional physics that further complicate modeling efforts. In this presentation, we perform three-dimensional Eulerian-Lagrangian simulations of particle-laden under-expanded jets to study the dynamics of inertial particles in compressible flows. The gas-phase equations are solved using a high-order, energy stable finite difference discretization. Particle trajectories and velocity distribution together with DMD modes are a high-order, energy stable finite difference discretization. A combined ghost-point / direct-forcing immersed boundary method is employed to model the nozzle geometry. The focus of the present work is to assess the capability to simulate particle dynamics through a series of compression and expansion waves (Mach diamonds). Particle trajectories and velocity distribution together with DMD modes are compared between different drag models and experiments. Two-way coupling effects on the structure of the Mach diamonds are also reported.

This work was funded by the Jet Propulsion Lab Center Innovation Fund. Computational resources provided by NASA Advanced Supercomputing.

11:00AM S37.00004 Particle Scale Heat Transfer Calculations and Flow Characteristics in a Fluidized Bed with Immersed Tube by CFD-DEM Approach

ANGELO VIGNOLO, Andrea Vigniolo, University of Insubria, Italy, COSIMO LIVI, GIANLUCA DI STASO, HERMAN J.H. CLERCX, Eindhoven University of Technology — The survival of fish passing through a hydroelectric power plant is a critical challenge to understand and address adverse environment impacts. Alternative routes to bypass fish in extreme hydraulic conditions are designed in many power plants, fish bypass still remains a critical issue. A computational fluid dynamics based tool, biological performance assessment (BioPA) method, has been developed to assess the biological performance of fish passage during the turbine design phase in new and existing turbine designs. The current work reports high fidelity turbulent flow investigations to predict the behavior, trajectories, and collisions of inertial particles in an idealized hydro-turbine model to predict the biological impact of hydro turbines on fish passage. Turbulent flow simulations were conducted using detailed eddy simulation to accurately capture the flow recirculation and wake region proximate to representative geometry of the distributor of a typical hydro turbine. The flow field proximate to the distributor geometry significantly impacts the trajectory and collision of particles. Naturally buoyant spherical and cylindrical particles were released at the upstream of the distributor. The trajectory and collision rate of the particles to distributor wall is computed and further compared with corresponding experimental results for different conditions.

11:20AM S37.00005 CFD tools for characterizing particle passage through an idealized hydro-turbine model

RAJESH SINGH, Pacific Northwest National Laboratory, PEDRO ROMERO-GOMEZ, Andritz HYDRO GmbH, 4030 Linz, Austria, MARSHALL RICHMOND, SAMUEL HARDING, WILLIAM PERKINS, Pacific Northwest National Laboratory — The survival of fish passing through a hydroelectric power plant is a critical challenge to understand and address adverse environment impacts. Alternative routes to bypass fish in extreme hydraulic conditions are designed in many power plants, fish bypass still remains a critical issue. A computational fluid dynamics based tool, biological performance assessment (BioPA) method, has been developed to assess the biological performance of fish passage during the turbine design phase in new and existing turbine designs. The current work reports high fidelity turbulent flow investigations to predict the behavior, trajectories, and collisions of inertial particles in an idealized hydro-turbine model to predict the biological impact of hydro turbines on fish passage. Turbulent flow simulations were conducted using detailed eddy simulation to accurately capture the flow recirculation and wake region proximate to representative geometry of the distributor of a typical hydro turbine. The flow field proximate to the distributor geometry significantly impacts the trajectory and collision of particles. Naturally buoyant spherical and cylindrical particles were released at the upstream of the distributor. The trajectory and collision rate of the particles to distributor wall is computed and further compared with corresponding experimental results for different conditions.

11:30AM S37.00006 An AMR moving cut-cell algorithm for particle-laden flows

ARTHUR GHIGO, Department of Mathematics, University of British Columbia, STEPHANE POPINET, Institut Jean le Rond d’Alembert — The capability to simulate a two-way coupled interaction between an arbitrary-shape colloidal particle and a fluid is important for a range of practical applications, however the computational cost can become burdensome. We propose a systematic error analysis of finite-size particle dynamics using the Hermite regularized Lattice Boltzmann method, for different particles resolution, addressing translational and rotational systems separately. The motivation is the understanding of the degree of accuracy provided by different boundary condition models at the fluid-solid interface, with a focus on cases where particles are discretized using few lattice grid points. We show that through a second-order accurate interpolated bounce-back scheme a strong error suppression can be achieved with respect to the standard first-order accurate approach, allowing to simulate smaller particles and thus improving computational performances. An improved version of the former interpolation scheme is developed in order to compute particle boundary position also when the particle is crossing a link between two fluid nodes, showing that the capability to resolve non-spherical particles is further enhanced.
12:02PM S37.00008 Qualitative Benchmarking of a Developmental CFD-DEM Code1

WILLIAM FULLMER2, JORDAN MUSSE, National Energy Technology Laboratory, ANN ALMGREN, MICHELE ROSSO, OSCAR ANTEPARA, ROBERTO PORCU, JOHANNES BLASCHKE, Lawrence Berkeley National Laboratory —

MFIX-Exa is a new code being developed by the National Energy Technology Laboratory and Lawrence Berkeley National Laboratory as part of the U.S. Department of Energy’s Exascale Computing Project. MFIX-Exa originated by combining the discrete element method (DEM) modules of the classic MFIX code (mfix.netl.doe.gov) with a modern low Mach number projection method for the continuous fluid phase. The new algorithm is implemented using the AMReX software framework for massively parallel block-structured applications (amrex-codes.github.io). This work utilizes a set of qualitative benchmarks which emphasize the phenomenology of a given problem rather than averaged statistical measures typically used in fluid-particle validation studies. The following cases are considered: 1) clustering of inelastic particles in the gas-solid homogenous cooling system, 2) granular RayleighTaylor instability of particles draining into a gas, 3) rapid injection of a single bubble into a particle bed at incipient fluidization, 4) ordered left-right bubbling pattern of a periodically fluidized bed, 5) segregation of “Brazil nuts” to the top of a small bed, 6) axial banding (segregation) of bi-disperse rotating tumbler. A range of agreement with the expected phenomena is observed and reported.

1This research was supported by the Exascale Computing Project (17-SC-20-SC), a collaborative effort of the U.S. Department of Energy Office of Science and the National Nuclear Security Administration.

2Leidos Research Support Team

Tuesday, November 26, 2019 10:31AM - 11:49AM —
Session S38 Porous Media Flow Mixing and Turbulence 620 - Xiaoliang He, PNPL

10:31AM S38.00001 Pore-scale Direct Numerical Simulation of Turbulent Flows through a Randomly-Packed Porous Medium1

XIAOLIANG HE, MARSHALL RICHMOND, WILLIAM PERKINS, TIMOTHY SCHEIBE, Pacific Northwest National Laboratory, SOURABH APTE, Oregon State University — Turbulent flows through randomly-packed porous media are ubiquitous in both natural and engineered systems. It is of great value to understand the important turbulence characteristics in the randomly packed beds of relevance for heat transfer applications in chemical/nuclear reactors, as well as some environmental problems such as hyperviscous exchange at the interface between free stream water and the underlying sediment. Currently, very few pore-scale direct numerical simulations (DNS) of high Reynolds number flows through packed beds have been conducted and almost all of these studies focused on structurally packed or dilute systems. In the present work, DNS are performed in a randomly-packed triply-periodic porous medium with a porosity of 0.37. The Eulerian and Lagrangian statistics of turbulence, TKE budget, anisotropy distribution in confined pore geometries are investigated. These observations are compared with our previous work on triple-periodic, face centered cubic (FCC), to understand the influence of the heterogeneity on the turbulent statistics. It is observed that the integral length scale in the random packing is larger than that from structured packing.

1The computation is performed using Cascade HPC at EMSL

10:44AM S38.00002 Transition to turbulence in randomly packed porous media: scale evolution of vortical structures

REZA M. ZIAZI, JAMES LIBURDY, Oregon State University — Vortical structures are the driving mechanism behind the process of transition to turbulence in randomly structured porous media that is observed in many natural processes; natural canopy forest fires and biological systems such as cardiovascular and respiratory. The investigation of vortex evolution during the onset of turbulence is performed by using time-resolved PIV to identify the flow structures, and measure the scale and energy of swirling structures as related to pore- and macro-scale Reynolds numbers. Objective local region-type (λ1) versus non-local (λ2) vortex identification methods are employed to detect the asymptotic scales at larger Re during transition from 100 to 1000. The direct measure of the size, strength, and number density of vortical structures are observed to show a similar trend and asymptote to turbulent scales at higher pore-scale Reynolds numbers. The shear and rotational contribution of vortical structures are influenced differently from pore- versus macro-scale Reynolds numbers which interprets the scale evolution during transition process.

10:57AM S38.00003 Dispersion and stretching in 3D Porous media

BLOEN METZGER, Aix-Marseille University, IUSTI-CNRS UMR 7343, 13453 Marseille, France, GEP TEAM, OSUR TEAM — Mixing processes in complex flows are governed by the dispersion and stretching induced by the flow in question. However, even for the apparently simple case of a viscous flow through an isotropic porous medium, the statistics of these crucial kinematics quantities remain uncertain. We experimentally tackle this problem using an index-matched porous medium composed of randomly packed solid spheres. The 3D Eulerian velocity-field is characterized with an unprecedented resolution. The dispersion of advective particles is measured and explained in terms of the velocity statistics. Last, the stretching laws are measured for the first time using a Lagrangian stretching reconstruction method. These results provide solid grounds for a full description of mixing processes in porous media.

11:10AM S38.00004 Geometric disorder regulates dispersion in viscoelastic porous media flows

DEREK M. WALKAMA, NICOLAS WAISSBORD, JEFFREY S. GUASTO, Tufts University — In this work, we study the dispersion of microparticles in viscoelastic fluid flow through model, microfluidic porous media, comprising arrays of either hexagonally ordered or randomly disordered pillars. Similar to previous work at high Peclet number, we show that a viscoelastic flow instability in the ordered medium enhances dispersion transverse to the mean flow direction with increasing Weissenberg number (Wi). In contrast, we demonstrate that geometric disorder has two main consequences for transport: First, disorder quenches the elastic instability and thus, suppresses transverse dispersion due to a lack of chaotic velocity fluctuations. Second, we observe an enhancement of longitudinal dispersion, which corresponds with the emergence of channelized flow within the disordered medium. These filamentous flows strengthen in intensity with increasing Wi, where the origin of the increased longitudinal transport stems from their strongly-correlated streamwise flow speed.
10:31AM S39.00001 Turbulence in Forced Stratified Exchange Flows. KATHERINE M. SMITH, JOHN R. TAYLOR, JAMIE PARTRIDGE, ADRIEN LEFAUVE, PAUL LINDEN, Department of Applied Mathematics and Theoretical Physics (DAMTP), University of Cambridge and The BP Institute, University of Cambridge — Continuously forced, stratified exchange flows occur in many geophysical systems, such as ocean basins, between continental shelves and the deep ocean, and at the mouth of rivers and estuaries. These exchange flows can be sensitive to instabilities that promote the growth of turbulence and increase mixing between the two differing flows. While these mixing processes are assumed to be important to global ocean budgets, they are unresolved within Earth system models and therefore must be fully understood in order to accurately parameterize the internal waves. This talk presents results from three-dimensional direct numerical simulations of stratified exchange flows that are continuously forced by weakly relaxing the buoyancy and streamwise velocity. The response of different domain geometries was investigated, and the results of simulations show that the slope of the side walls dramatically affects the surface mixed layer. Mixing is enhanced and penetrates deeper in the vertical as slope inclination grows. Under resonant forcing, surface shear stress and resonant frequencies of the internal waves. The response of different domain geometries was investigated, obtained varying the inclination angle of the side walls, from zero (vertical walls) to 90°. When the frequency of the forcing is close to the first mode, resonant internal seiche occurs, in other cases forcing conditions are non-resonant. The results of simulations show that the slope of the side walls dramatically affects the surface mixed layer. Mixing is enhanced and penetrates deeper in the vertical as slope inclination grows. Under resonant forcing, surface waves are much more energetic and there is less change in the density profile compared to non-resonant conditions.

10:44AM S39.00002 Large Eddy Simulation of the response of a stratified reservoir with inclined walls to an oscillating surface shear stress. SARA MARKOVIC, VINCENZO ARMENIO, Dipartimento di Ingegneria e Architettura, Università di Trieste, 34127 Trieste, Italy — We present results of numerical simulations of the response of a reservoir with two-layer stratification to an oscillating surface shear stress. The simulations are carried out at a laboratory scale, using Large Eddy Simulation. We solve the three dimensional Navier-Stokes equations under the Boussinesq approximation for the density field using a model based on buoyantBoussinesqPimpleFoam implemented in the OpenFOAM library. The model was validated by reproducing experimental results for reservoir response to surface shear stress and resonant frequencies of the internal waves. The response of different domain geometries was investigated, obtained varying the inclination angle of the side walls, from zero (vertical walls) to 90°. When the frequency of the forcing is close to the first mode, resonant internal seiche occurs, in other cases forcing conditions are non-resonant. The results of simulations show that the slope of the side walls dramatically affects the surface mixed layer. Mixing is enhanced and penetrates deeper in the vertical as slope inclination grows. Under resonant forcing, surface waves are much more energetic and there is less change in the density profile compared to non-resonant conditions.

10:57AM S39.00003 Interaction between an inclined gravity current and a pycnocline in a two-layer stratification. YUKINOBU TANIMOTO, NICHOLAS OUELLETTE, JEFFREY KOSEFF, Department of Civil and Environmental Engineering, Stanford University — A series of laboratory experiments were conducted to investigate the characteristics of a dense gravity current flowing down an inclined slope into a quiescent two-layer stratification. The presence of the pycnocline causes the gravity current to split and intrude into the ambient at two distinct levels of neutral buoyancy, as opposed to the classical description of gravity currents in stratified media as being either a pure underflow or interflow. The splitting behavior is observed to be dependent on the Richardson number (Ri) of the gravity current, formulated as the ratio of the excess density and the ambient stratification. For low Ri, underflow is more dominant, while at higher Ri interflow is more prominent. As Ri increases, however, we find that the splitting behavior eventually becomes independent of Ri. Additionally, we have also identified two different types of waves that form on the pycnocline in response to the intrusion of the gravity current. An underflow-dominated regime causes a pycnocline displacement where the speed of the wave crest is locked to the gravity current, whereas an interflow- dominated regime launches an internal wave that moves much faster than the gravity current head or interfacial intrusion.

1NSF
11:10AM S39.00004 Modelling stratified wall-bounded turbulence using resolvent analysis1, MUHAMMAD AHMED, ANDREW THOMPSON, BEVERLEY MCKEON, California Institute of Technology — The effects of an active scalar have on incompressible wall-bounded turbulence are investigated using the resolvent framework (McKeon & Sharma, 2010, JFM). The state of the flow system is expressed as the result of applying the linear resolvent operator to the nonlinear terms in the governing Navier-Stokes equations with the Boussinesq approximation. To investigate the relationship between velocity and scalar fluctuations, the formulation is extended to include a scalar equation (Dawson et al., 2018, AIAA) and a scalar component acting in the wall-normal direction in the momentum equations. It is found that the Richardson number has a significant effect on the shape and phase of the velocity and scalar modes across the critical layer. In addition, it is shown that active scalar modes have a significant impact on the energy transfer between velocity mode components at varying scales. Furthermore, we visualise mode combinations that are representative of coherent structure observed in the atmosphere and oceans to gain a better understanding of the spatial wavenumber spectrum observed in nature.

1This research is funded by the Office of Naval Research through Grant ONR N00014-17-1-3022 to the California Institute of Technology

11:23AM S39.00005 Large-eddy simulations of stratified turbulence: an anisotropic subgrid-scale closure, SINA KHANI, University of Washington, MICHAEL WAITE, University of Waterloo — The horizontal and vertical grid spacings are generally equal in large-eddy simulations (LES) of stratified turbulence. In simulations of larger-scale motions, however, it is computationally affordable to use different grid spacings in the horizontal and vertical. In this talk, we introduce a new subgrid-scale (SGS) parameterizations based on horizontal filtering of equations of motions in stratified turbulence. The horizontal and vertical SGS dissipations and fluxes are determined based on a Reynolds decomposition of the flow. Our model can successfully reproduce the flow characteristics, moments and parameters, such as time series of kinetic and potential energy, horizontal and vertical wavenumber energy spectra and mixing efficiency, similar to those in direct numerical simulations while the computational cost is considerably reduced in LES. We suggest that our new SGS model can also improve the dissipative performance of the horizontal Smagorinsky closure in current atmospheric and oceanic models without adding any ad hoc energizing terms at smaller scales.

11:36AM S39.00006 Wall shear stress fluctuations induced by shallow-water Langmuir turbulence, BINGQING DENG, ANQING XUAN, LIAN SHEN, University of Minnesota — In shallow water with the existence of surface waves, large-scale full-depth Langmuir circulations can be generated due to the interaction between wind-driven currents and the Stokes drift of shallow-water waves. We perform wall-resolved LES of the Craik-Leibovich equation to study shallow-water Langmuir turbulence. It is found that the full-depth Langmuir circulations directly leave large-scale footprint near the water bottom and hence have significant contributions to the wall shear stress fluctuations. The full-depth Langmuir circulations also alter the distribution of other coherent structures, so that the distribution of the wall shear stress fluctuations contributed by other coherent structures is different from the wall turbulence in the absence of Stokes drift of surface waves. The magnitude of the wall shear stress fluctuations at various locations is predicted by the velocity fluctuations induced by the full-depth Langmuir circulations.

11:49AM S39.00007 DNS of stratified Ekman layers over rough surfaces, SUNGWON LEE, IMAN GOHARI, SUTANU SARKAR, University of California, San Diego, UNIVERSITÉ DE CALIFORNIE, SAN DIEGO TEAM — We investigate the evolution of stratified Ekman boundary layers in the presence of surface roughness. The roughness elements are 2-dimensional bumps and the neutral flow is in the transitional rough regime. A cooling buoyancy flux which is applied for a finite time period is responsible for the stabilizing stratification. The Reynolds number is moderate so that a broad parametric study of the influence of roughness height ($h^+$) and cooling flux (normalized Obukhov length, $L^+$) can be performed using direct numerical simulation (DNS). A cooling flux corresponding to $L^+ \approx 700$ is sufficient to cause the initial collapse of turbulence for both smooth and rough surfaces. Buoyancy and the slope of the surface roughness elements act in conjunction to affect the state of boundary-layer turbulence after the initial transient. The final value of the bulk Richardson number ($Ri_b$), which is a function of both $L^+$ and roughness properties, is found to provide guidance on the overall state of the flow, e.g., weak or strongly stable in the sense of Mahrt (1998); continuous, globally intermittent or locally intermittent.

12:02PM S39.00008 Multiphase plumes in a rotating environment, DARIA FRANK, University of Cambridge, JULIEN LANDEL, University of Manchester, STUART DALZIEL, PAUL LINDEN, University of Cambridge — The Deepwater Horizon (DwH) blowout in 2010 resulted in a formation of a multiphase plume consisting of oil droplets and gas bubbles. The multiphase component of a plume is commonly characterised by the so-called slip velocity. Additionally, the duration of the DwH spill of several months implies that the plume is likely to have been affected by the Earth’s rotation as was conjectured by previous numerical and experimental studies. The complex interplay between the slip velocity of the multiphase effluent and the system rotation as well as their combined effects on the internal plume dynamics and the associated subsurface dispersion of pollutants are still poorly understood. Yet, characterising these effects is important for oil spill mitigation purposes. In this talk, we present results from small-scale laboratory experiments on bubble plumes released into a rotating environment that were conducted for a wide range of Rosby numbers and several bubble slip velocities. By performing a rigorous image analysis, we focus on three particular aspects of the problem: the evolution of the plume structure during its initial rise, the subsurface dispersion of the effluent once the plume has risen through the entire water column and the lateral spreading of the plume on the water surface.
10:31AM S40.00001 Wall-bounded turbulence control using a Monte-Carlo approach
1, OSCAR FLORES, ROBERTO PASTOR, Universidad Carlos III de Madrid, ALBERTO VELA-MARTIN, Universidad Politécnica de Madrid — Wall-bounded turbulence is very important in engineering applications involving fluids. Indeed, wherever flow control is sought in engineering applications both sensing and actuation are restricted to devices placed at solid walls. In the present work we tackle the classical problem of skin friction control in wall-bounded turbulence for the case of localised actuation using a Monte Carlo approach. To that end, DNS of a minimal channel of the buffer layer (Re = 180) are run using GPUs. The actuation is a volumetric vertical force applied close to the wall on a characteristic volume L3 and with duration T. Several forcing are considered, with L = [50, 100] and T = [25, 50, 100]. Their effect is evaluated for O(104) episodes, directly comparing the instantaneous skin friction of forced and unforced simulations. The statistical analysis shows that drag increase and decrease are equally probable when the forcing is randomly applied. Episodes with drag decrease show a positive (negative) vertical force being applied to passing sweep (ejection) events, in agreement with opposition control strategies. Preliminary results also suggest that a skin friction sensor upstream of the forcing is a better trigger for the forcing than a wall-pressure sensor at the same location.

1 Funded by the Coturb program of the European Research Council

10:44AM S40.00002 Selective opposition-like control of large-scale structures in wall-bounded turbulence
2, JOSEPH IBRAHIM, University of Cambridge, ANNA GUSEVA, School of Aeronautics, Universidad Politécnica de Madrid, RICARDO GARCIA-MAYORAL, University of Cambridge — We investigate the effect of controlling large-scale, logarithmic-layer turbulent structures, which have a characteristic size and aspect ratio that scale with the distance from the wall. The aim is to quantify the effect of suppressing these structures while leaving the near-wall turbulent dynamics unaltered. We conduct direct numerical simulations of turbulent channel flows at Re ≈ 500–1000 and artificially remove certain streamwise and spanwise wavelengths of the wall-normal velocity across a range of heights. The wavelengths chosen depend on the target height (and size) of the structures that we wish to target. Our preliminary results suggest that the reduction in drag is observed as a positive, outward shift in the mean velocity profile above the target height that scales in outer units.

2This work has been partially supported by the Coturb program of the European Research Council and the Engineering and Physical Sciences Research Council (EPSRC), UK.

10:57AM S40.00003 Direct Numerical Simulation of oscillatory boundary layers in the intermittently turbulent regime: coherent structures, laminarization and scaling
, DIMITRIOS K. FYTANIDIS, JOSE M. MIER, MÁRCHELO H. GARCIA, PAUL FISCHER, University of Illinois at Urbana-Champaign — Experiments conducted at the Ven Te Chow HydroSystems Laboratory (UIUC) in the transitional regime of oscillatory boundary layer flows with smooth bed, show changes in the phase shift between bed shear stress and free-stream velocity maxima (Mier J.M., 2015). Nevertheless, limited by the point-wise measurements (Laser Doppler Velocimetry), it was not possible to relate this finding with the development of three-dimensional flow structures. In this work, Nek5000 is used to perform Direct Numerical Simulation (DNS) of oscillatory boundary layer flows in moderately high Re numbers. DNS results of mean flow and turbulent statistics compare well with experimental observations. Coherent structures and their effect on turbulence characteristics are examined. Vortex tubes have minimal effect on turbulent statistics and friction factor, while turbulent spots defined as sporadic, highly-energetic, lambda-shaped structures, have a significant effect. Analysis of phase shift between free-stream velocity and bed shear stress maxima agrees with the experimental observations. Theoretical analysis is performed for the prediction of laminarization during acceleration phase. A generalized logarithmic law is proposed for accelerating flows using a novel composite acceleration-shear velocity scale.

11:10AM S40.00004 The interfaces with the freestream of a spatially developing boundary layer
1, JAMES WALLACE, University of Maryland, XIAOHUA WU, Royal Military College of Canada. JEAN-PIERRE HICKEY, University of Waterloo — We have investigated the interfaces with the freestream turbulence of the laminar boundary-layer (LBFTI), turbulent spots (TSFTI) and the turbulent boundary-layer (BFITI) using direct simulation of a zero pressure-gradient, smooth-wall boundary-layer flow developing from a laminar state, through transition, to a developed turbulent state. Probability density functions of temperature and its derivatives are used to select the interface identification thresholds. These interfaces are confirmed to be physical by the distinctive quasi-step-jump behavior in the swirling strength and temperature statistics along traverses normal to the BFITI and TSFTI. No interface normal inflection is detected across the LBFTI for swirling strength, temperature, vorticity magnitude, Reynolds shear stress, streamwise and normal velocity or turbulent kinetic energy. This casts serious doubt on the shear-sheltering hypothesis/theory which asserts that freestream fluctuations are blocked by the LBFTI. In the early stage of transition, quasi-spanwise structures exist on the LBFTI. The TSFTI shape is dominated by head prints of concentrated hairpin vortices. Further downstream, the BFITI geometry is strongly modulated by groves of hairpin vortices.

1Supported by the Natural Science and Engineering Research Council of Canada and Computer Canada

11:23AM S40.00005 Multi-structure turbulence in a boundary layer interacting with a uniformly sheared flow
1, STAVROS TAVOULARIS, CURTIS LIVINGSTON, University of Ottawa — Turbulence generated by two or more distinct production mechanisms and having two or more types of large-scale structure has been termed multi-structure. Depending on the conditions, such flows may retain a non-canonical character or relax to a canonical flow. This study investigates a case in which both production mechanisms persist as the flow evolves. It examines the multi-structure-turbulence region of a turbulent boundary layer (TBL) developing along a smooth wall in a water tunnel, while being adjacent to a nearly homogeneous, uniformly sheared flow (USF). Detailed measurements are collected with laser Doppler velocimetry, particle image velocimetry and hot film anemometry. The mean shear and the turbulent shear stress change direction from the TBL to the USF and vanish near the TBL edge. The statistical properties of the turbulence in the multi-structure region have been measured and compared to those in canonical TBL and USF. Particular interest focusses on the variation of the dissipation parameter across the flow and the shapes and orientations of the coherent structures in the multi-structure region.

1Supported by the Natural Sciences and Engineering Research Council of Canada.
benchmarking against conventional grid and random sampling methods. Best practices are highlighted for a class of generic fluid dynamics optimisation strategy and demonstrate its benefits to industrial simulation practitioners, including a reduction in design lead times, by minimising gelling under engineering constraints on both flow and geometric properties. We achieve computational tractability via a surrogate parameter dimensions. In this study, we investigate industrial gelling phenomena for non-Newtonian fluids in eccentric annuli. The task is to maximising an engineering performance metric. Here, simple strategies such as grid searching do not scale well with the number of optimisation variables. The trade-off for this accuracy, however, can be prohibitively high computational cost. These costs are further compounded when multiple calls to an expensive simulation code are required, for example when determining the optimal parameters for manufacturing an engineering performance metric. Here, simple strategies such as grid searching do not scale well with the number of optimisation parameter dimensions. In this study, we investigate industrial gelling phenomena for non-Newtonian fluids in eccentric annuli. The task is to minimise gelling under engineering constraints on both flow and geometric properties. We achieve computational tractability via a surrogate model optimisation strategy and demonstrate its benefits to industrial simulation practitioners, including a reduction in design lead times, by benchmarking against conventional grid and random sampling methods. Best practices are highlighted for a class of generic fluid dynamics optimisation problems.

1Royal Academy of Engineering; PETRONAS; EPSRC, UK; AI for Science and Government (Data-Centric Engineering Programme, The Alan Turing Institute, UK). Imperial College Research Fellowship for IP.

10:36AM S40.00006 Development of a turbulent boundary layer subjected to free-stream turbulence

YANNICK JOOSS, LEON LI, TANIA BRACCHI, R. JASON HEARST, Norwegian University of Science and Technology — Turbulent boundary layers are a fundamental flow that exists in a wide range of natural processes and technical applications. Over the past three decades the effect of free-stream turbulence on the features of a canonical zero-pressure-gradient turbulent boundary layer has been studied extensively, with particular focus on single downstream positions. Nonetheless, there has been little attention given to the influence of varying free-stream turbulence intensity $u'_\infty/U_\infty$ on the actual streamwise development of a boundary layer. This study addresses this gap with hot-wire measurements in a water channel. Free-stream turbulence is created and varied with an active grid. Wall-normal boundary layer scans are performed along the centerline of the channel at multiple streamwise positions. The resulting mean velocity profiles and turbulent fluctuations in the boundary layer for four different free-stream turbulence conditions are analyzed. Further insight is gained by looking at the spectral distribution of energy at selected streamwise positions.

Tuesday, November 26, 2019 10:31AM - 12:15PM
Session S41 CFD: General II
6c - Omar Matar, Imperial College London

10:31AM S41.00001 Surrogate-model Optimisation Strategies for non-Newtonian Gelling1

ADAM KUTNAR, Imperial College London, LACHLAN MASON, Alan Turing Institute, UK, INDRANIL PAN, RICHARD CRASTER, OMAR MATAR, Imperial College London — Highly resolved simulations have advanced to the extent that they are routinely relied upon in engineering design. The trade-off for this accuracy, however, can be prohibitively high computational cost. These costs are further compounded when multiple calls to an expensive simulation code are required, for example when determining the optimal parameters for manufacturing an engineering performance metric. Here, simple strategies such as grid searching do not scale well with the number of optimisation parameter dimensions. In this study, we investigate industrial gelling phenomena for non-Newtonian fluids in eccentric annuli. The task is to minimise gelling under engineering constraints on both flow and geometric properties. We achieve computational tractability via a surrogate model optimisation strategy and demonstrate its benefits to industrial simulation practitioners, including a reduction in design lead times, by benchmarking against conventional grid and random sampling methods. Best practices are highlighted for a class of generic fluid dynamics optimisation problems.

1We acknowledge RCGI-USP, FAPESP (2014/50279-4) and Shell Brasil.

10:44AM S41.00002 Geometric parameter optimization of a liquid jet liquid ejector1

IVAN KORKISCHKO, CECCO, Nuclear and Energy Research Institute, Sao Paulo, Brazil, FELIPE SILVA MAFFEI, Dept. Mechanical Engineering, POLI, University of Sao Paulo, Sao Paulo, Brazil, RAFAEL DOS SANTOS GIORIA, Dept. Mining and Petroleum Engineering, POLI, University of Sao Paulo, Sao Paulo, Brazil, JULIO ROMANO MENEGHINI, Dept. Mechanical Engineering, POLI, University of Sao Paulo, Sao Paulo, Brazil — Ejectors are devices employed as pumps or compressors, which work transferring momentum from a primary fluid (high pressure) to a secondary fluid (low pressure). On the one hand, their main advantages over standard pumps and compressors are no moving parts, no need of lubricants and seals, and low noise and maintenance. On the other hand, ejectors have low efficiency compared to other devices and a very narrow region of optimal operation. Thus, ejectors certainly benefit from optimization studies. This investigation was based on a CFD model of a liquid jet liquid (LJL) ejector. The finite element method was used, coupled with the k-epsilon turbulence model. The optimization study had three steps. First, the constants of the turbulence model were recalibrated to minimize the difference between the numerical and experimental efficiency curves. Second, using the main geometric parameters as control variables, the peak efficiency was maximized. Finally, the optimized geometry was further improved, considering the transitions between the different ejector components, which were originally sharp corners. The optimized round corners increased the ejector efficiency.

1Helpful discussions with Andrew Russel and Prof. Paul Luckham of Imperial are gratefully acknowledged as is funding via a PETRONAS/Royal Academy of Engineering Research Chair for OKM.

10:57AM S41.00003 Direct Numerical Simulations of Mixing: From Aeration Tanks to Food Mixers1

SEUNGWON SHIN, Hongik University, South Korea, LYES KAHOUADJI, Imperial College London, JALEL CHERGUI, DAMIR JURIC, LIMSI, CNRS, France, RICHARD CRASTER, OMAR MATAR, Imperial College London — The dynamics of stirred tanks has been studied over a wide range of laboratory and industrially conditions and scales, in order to improve the mixing efficiency. We use a three-dimensional two-phase flow dynamics solver coupled with a direct Forcing Method to handle accurately the fluid structure interaction occurring in any of the stirred tank. The numerical framework employed here circumvents numerous meshing issues normally associated with constructing complex geometries (impellers, baffles, etc.) within typical computational fluid dynamics packages. All these solid structures are constructed via a module that defines solid objects by means of a static distance function. Typical examples will be presented such us aeration (bubbling mixing), cavern formation (stirring viscoplastic fluids), and typical Egg Beaters with different types of fluid rheology.

1Helpful discussions with Andrew Russel and Prof. Paul Luckham of Imperial are gratefully acknowledged as is funding via a PETRONAS/Royal Academy of Engineering Research Chair for OKM.

11:00AM S41.00004 Numerical Simulation of Crude-Oil Fouling with the Volume-of-Fluid Method1

GABRIEL GONCALVES, Imperial College London, MIRCO MAGNINI, University of Nottingham, OMAR MATAR, Imperial College London — In oil production or processing pipelines, the change in thermodynamical conditions may cause components of crude-oil to precipitate and adhere to the wall, leading to undesired changes in the hydraulics and thermal performance of the system over time. Although significant progress has been achieved in understanding qualitatively the main mechanisms of wax formation and removal, quantitative comparisons with experimental data are still heavily dependent on calibration to experimental data. In order to perform high-fidelity simulations in arbitrary geometries, a two-phase multi-component solver with heat transfer and phase change was implemented in the OpenFOAM open-source framework. The volume-of-fluid method is used for tracking the interface between fouling layer and liquid phase. The implementation was compared with previous calculations performed in a commercial CFD platform and preliminarily validated with experimental data from the literature.

1CNPq funding for GG, and PETRONAS/Royal Academy of Engineering Research Chair for OKM is gratefully acknowledged.
11:23AM S41.00005 Three-dimensional Numerical Simulations of Turbulent Surfactant-laden Jets, JALEL CHERGUI, LIMSI, CNRS, France, RICARDO CONSTANE-AMORES, ASSEN BATCHVAROV, LYES KAHOUADJI, Imperial College London, SEUNGWON SHIN, Hongik University, South Korea, DAMIR JURIC, LIMSI, CNRS, France, RICHARD CRASTER, OMAR MATAR, Imperial College London — Liquid atomisation processes are widely used to break down a liquid stream into smaller droplets to enhance its mixing with a stagnant phase. These streams may be contaminated with surfactants, whose concentration variations lead to gradients in surface tension and associated Marangoni stresses. Here, we study, for the first time, the effect of surfactant on the complex interfacial dynamics associated with a turbulent jet. We use a hybrid front-tracking/level-set method to capture the dynamics of the complex topological changes in this flow. The numerical method allows the natural tracking of the concentration of interfacial surfactant species and the faithful modelling of its spatio-temporal evolution. Our model also accounts for surfactant solubility and bulk-interface mass exchange. We perform a full parametric study of the effect of surfactant properties on the dynamics. The effect of Marangoni stresses is analysed in terms of the mechanisms giving rise to the droplet size distributions depending on the elasticity number. An attempt to understand the interaction between the observed vortical structures accompanying the flow and the regions of elevated surfactant concentration will also be presented.

1Funding from BP, EPSRC, PETRONAS, Royal Academy of Engineering (Research Chair for OKM).

11:36AM S41.00006 Computational Investigation of the Retropulsive Jet Produced by Antral Contraction Waves in a Model Stomach, KATHLEEN FEIGL, Michigan Technological University, SAMER ALOKAILY, University of Petra, FRANZ TANNER, Michigan Technological University — Numerical simulations are performed to investigate the retropulsive jet that is produced by peristaltic motion during the mixing and digestion process in a model human stomach. The geometrical model for the stomach consists of an axisymmetric conical-shaped tube with a wall at one end which represents the antrum and closed pylorus. The antral contraction waves which produce the peristaltic flow, and consequently the retropulsive jet near the closed pylorus, are modeled as traveling waves along the tube boundary which deform the computational mesh. This geometrical model and the boundary deformation algorithm are implemented into a C++ library and then coupled with the open source code OpenFOAM. The effect of various parameters on the retropulsive jet near the pylorus is investigated. These parameters include the fluid viscosity, wave speed, wave width and maximum relative occlusion. The retropulsive jet is quantified by its peak velocity and jet length along the centerline at maximum relative occlusion. For each wave geometry, it is found that the velocity and pressure curves scale with wave speed at low Reynolds numbers. Between different wave geometries, scaling laws are proposed and tested for the peak centerline velocity and jet length at low Reynolds numbers.

11:49AM S41.00007 Hydrodynamics of solid objects impacting on free surface fluid, DEEPAK KUMAR PANDEY, HEECHANG LIM, Pusan National University — The impact studies of different shape and size of solid objects and curved plates on the free surface of fluid have a great practical significance from engineering perspectives. It is worth noting here that such type of impact phenomena can be correlated with slamming of ship bow, seaplanes, submarines, military projectiles, etc. in the seawater. The objectives of this paper are to experimentally and numerically analyze the field variables (pressure distribution, splashing, wave propagation, etc.) arising out of free fall of different shape and size of solid objects (square prism, rectangular prism, triangular pyramid, cylinder, sphere, etc.) on the free surface of the fluid. The fluids employed were water and oils having different viscosity. During the experiments, free-falling solid objects as projectiles were allowed to impact the free surface of the fluid. The height and the angle of the free falling solid objects were also varied in order to observe its effect on splashing and wave propagation. The interface behaviors and wave propagation have been captured using a high-speed camera. However, pressure sensors and strain gauges were employed in order to record the pressure and impact load at different locations. The simulation results validate the experimental results.

12:02PM S41.00008 Active learning methodologies for surrogate model development in CFD applications, INDRANIL PAN, GABRIEL GONCALVES, ASSEN BATCHVAROV, YUXIN LIU, YUJI LIU, VIK-NESWARAN SATHSIVAM, NICHOLAS YIAKOMI, Imperial College London, LACHLAN MASON, Alan Turing Institute, OMAR MATAR, Imperial College London — Computational fluid dynamic simulations typically have high computational costs, such that for parametric analysis and engineering design an inexpensive surrogate model, which is capable of reproducing the trends of some variables of interest, may be desired. However, generating regressions based on a full grid-based parametric variation is generally infeasible even for a system with moderate number of parameters. In this work, a wide array of active learning techniques were coupled with different regression models to achieve high predictive performance under the constraints of a limited function evaluation budget. The case studies involve flows of industrial relevance and the results outline some best practices for such simulations and highlights future research directions.

1CNPq funding for GG, and PETRONAS/Royal Academy of Engineering Research Chair for OKM is gratefully acknowledged.