8:00AM AA.00001 LES of scalar transport in wave and wind-driven flows with large-scale structures1 CIGDEM AKAN, ANDRES TEJADA-MARTINEZ, University of South Florida, CHESTER GROSCH, Old Dominion University — Near-surface scalar (mass) transport results from large-eddy simulation (LES) of wind-driven flow with and without full-depth Langmuir circulation (LC) are reported. LC is generated by wave-current interaction and consists of counter rotating vortices aligned in the direction of the wind. LES driven by wind and wave forcing conditions measured during field observations of full-depth LC by Gargett and Wells (Journal of Fluid Mechanics, 576, 27-61, 2007) shows that this large-scale, downwind-elongated structure increases mass transfer velocity (a measure of mass transfer efficiency) by approximately 60 percent with respect to a similar flow without surface wave effects (i.e. without LC). The LES will be used to test the accuracy of surface renewal-based parameterizations (models) in predicting surface transfer velocity increase in flows with LC. Statistical analysis of LES variables will be presented demonstrating that full-depth LC dominates near-surface mixing as well as transport somewhere else in the water column. In the absence of LC, near-surface small eddies contribute significantly towards the mass transport at the air-water interface.

8:13AM AA.00002 Large-eddy simulation of oxygen transport and depletion in waterbodies , CARLO SCALO, UGO PIOMELLI, LEON BOEGMAN, Queens University — Dissolved oxygen (DO) in water plays an important role in lake and marine ecosystems. Agricultural runoff may spur excessive plant growth on the water surface; when the plants die they sink to the bottom of the water bodies and decompose, consuming oxygen. Significant environmental (and economic) damage may result from the loss of aquatic life caused by the oxygen depletion. The study of DO transport and depletion dynamics in water bodies has, therefore, become increasingly important. We study this phenomenon by large-eddy simulations performed at laboratory scale. The equations governing the transport of momentum and of a scalar (the DO) in the fluid are coupled to a biochemical model for DO depletion in the permeable sediment bed [Higashino et al., Water Res. (38) 1, 2004)], and to an equation for the fluid transpiration in the porous medium. The simulations are in good agreement with previous calculations and experiments. We show that the results are sensitive to the biochemical and fluid dynamical properties of the sediment, which are very difficult to determine experimentally.

8:26AM AA.00003 Decay of a Passive Tracer in Two-Dimensional Turbulence-Application to Infinitely Fast Chemistry , FARID AIT CHAALAL, PETER BARTELLO, MICHEL BOURQUI, McGill University, Montréal, Canada — We investigate the effect of diffusion ($\kappa$) on chemical production in a 2D turbulent flow using ensembles of direct numerical simulations (DNS). Assuming an infinitely fast chemistry between two initially unmixed reactants, the problem simplifies to studying the mean absolute value of the passive tracer $\phi$ defined as the difference between the concentrations of the two reactants. The reaction speed is dictated by the diffusive flux across an isoline of $\phi$. The DNS show that production scales like $\kappa^2(t)$ where $p(t)$ is a positive decreasing function of time, during an initial transient period characterized by the exponential lengthening of the contact line. We show that the behavior of $p(t)$ is determined by the initial gradients along the contact line, with sharp gradients decreasing the effect of diffusion. The long time decay of the first moment of $\phi$ is exponential. At large Peclet number, the decay rate converges and the reaction speed is merely determined by the mixing. At small Peclet number ($\lesssim 5000$), the decay rate scales like $\kappa^{1/2}$, reflecting a purely diffusive behaviour.

8:39AM AA.00004 DNS of mixing in spatially developing shear layers , ANTONIO ATTILI, FABRIZIO BISSETTI, King Abdullah University of Science and Technology — The majority of models proposed for turbulent combustion rely on the knowledge of the mixture fraction $Z$, its pdf, the scalar dissipation, and, in some cases, a mixing time scale. Turbulent mixing in technical devices is almost always inhomogeneous and the flow regime is transitional so that spatially developing mixing layers are highly relevant. DNS have been performed for a Reynolds number up to $2 \times 10^6$, enough to identify mixing transition. All relevant statistics and their dependence on Reynolds number have been analyzed. Previous results show that the scalar pdf can be: non-marching, when the most probable value of the mixture fraction of mixed fluid is invariant across the width of the layer; marching, for which the most probable value is the mean mixture fraction at the location considered. The pdf behavior is discussed in several previous works and it has been found to be related to the Reynolds number and the level of turbulence development. The DNS results we present clearly show that the pdf evolves from non-marching to marching during the streamwise evolution of the scalar field.

8:52AM AA.00005 Dynamic model for the joint scalar probability in multi-species turbulent mixing1, J.R. RISTORCELLI, J. BAKOSI, Los Alamos National Laboratory — We present a probability density function (PDF) model for multi-species scalar mixing in turbulent flows. In the proposed model the scalars are governed by a system of stochastic differential equations, discretized and integrated in a Monte-Carlo fashion. The model is local in composition space, accounts for different scalar mixing rates and Schmidt numbers and can represent a variety of PDF shapes, including a multiple-delta in the unmixed and a joint (bounded) Gaussian in the fully mixed states. The method is intended for passive, active or reacting scalars in shear-driven and/or variable-density pressure-gradient-driven turbulence.

9:05AM AA.00006 Stochastic shell mixing model for scalars in homogeneous turbulence, YANJUN XIA, LANCE COLLINS, Cornell — We present a stochastic shell mixing model (SSMM) based on the eddy damped quasi-normal Markovian (EDQNM) theory to study turbulent mixing of scalars. The formulation combines the strengths of spectral model and the probability density function (PDF) method. The model addresses the scalar concentration of notional particles that is distributed across spectral shells using a Monte Carlo algorithm. The variance of the concentration within each shell reproduces the scalar spectrum at the corresponding wavenumber, and the sum of the concentrations over the shell for each particle provides the fine-grained joint PDF. Monte Carlo schemes in general have high accuracy for the solution of the scalar by the initial and boundary conditions. We present a special procedure for keeping the particle concentrations within the bounds. Due to the inherent full description of length and time scales, the SSMM model is capable of capturing the dependence of mixing on the molecular diffusivity, and hence the effects of differential diffusion. Results are compared with direct numerical simulations (DNS) and are in good agreement.
9:18AM AA.00007 Modeling Scalar variance from Direct Numerical Simulations of a turbulent mixing layer, BAPTISTE RAVINEL, ENSTA ParisTech, GUILLAUME BLANQUART, California Institute of Technology — Many studies have focused on analyzing and predicting the mixing of a scalar such as fuel concentration in turbulent flows. However, the subfilter scalar variance in Large Eddy Simulations (LES) still requires additional considerations. The present work aims at obtaining results for the turbulent mixture of a scalar in configurations relevant to reactive flows, i.e. in the presence of mean velocity/scalar gradients. A Direct Numerical Simulation (DNS) of a turbulent mixing layer has been performed by initially combining two boundary layers. The high order conservative finite difference low Mach number NGA code was used together with the BQuick scheme for the transport of mixture fraction. The self-similar nature of the flow and energy spectra have been considered to analyze the turbulent flow field. High order velocity schemes (4th order) were found to play an important role in capturing accurately the mixing of fuel and air. The scalar variance has been calculated by filtering the solution and has been compared to various models usually used in LES. Following an earlier study by Balarac et al. [Phys. Fluids 20 (2008)], the concept of optimal estimators has been considered to identify the set of parameters most suitable to express the subfilter variance. Finally, the quality of the standard dynamic approach has been assessed.

9:31AM AA.00008 Numerical diffusivity of scalar transport schemes in high Schmidt number flows, SIDDHARTHA VERMA, GUILLAUME BLANQUART — Accurate simulation of scalar transport in high Schmidt number turbulent flows is essential to studying pollutant dispersion, weather, and several oceanic phenomena. Experiments over such large scales are difficult to conduct and simulations often provide a more practical alternative. Scalar transport in turbulent flows is governed by Batchelor's theory which requires further validation for high Schmidt numbers and high Reynolds numbers. The present study focuses on the impact of numerical diffusivity of various commonly used schemes in the turbulent transport of high Schmidt number scalars. This analysis is performed first in laminar flows, including advection of a Gaussian distribution in steady flow field and scalar stretching in Taylor vortex configuration. Then, the study focuses on the ability of the transport schemes to reproduce faithfully turbulent flow fields with high accuracy and low dissipation while maintaining physical boundedness. This analysis is conducted for isotropic, homogeneous forced turbulence and a mixing layer. Finally, a semi-Lagrangian scheme is considered and its performance is analyzed in the limit of zero diffusion.

9:44AM AA.00009 Entanglement rules for random mixtures, EMMANUEL VILLERMAUX, Aix-Marseille Universite, IRPHE, JÉRÔME DUPLAT, Aix-Marseille Universite, IUSTI — We discuss how two subparts of a randomly stirred scalar mixture interact to form the overall concentration distribution. We derive in particular the appropriate composition laws in absence, and in the presence of strong correlation between the fields. The resulting concentration distributions compare favorably with several distinct experiments, illustrating the two limits (Phys. Rev. Letters 105, 034504, (2010)). The initial relative spatial position of the subparts plays a crucial role on the nature of their subsequent entanglement.

9:57AM AA.00010 Linking spectra and geometry in scalar fields, MIHKEL KREE, Aix-Marseille Universite, IRPHE, JÉRÔME DUPLAT, Aix-Marseille Universite, IUSTI, EMMANUEL VILLERMAUX, Aix-Marseille Universite, IRPHE — The spectral signature of a turbulent scalar field is usually interpreted in terms of cascades, leading to decaying power laws either in the inertial range (the −5/3 slope), or in the viscous subrange (the −1 slope). An intermittent scalar field produced by an elongational velocity field, whatever it may be, also gives rise to a −1 power law. We study here the mixing and homogenization of a passive scalar injected from a point source by both confining it in a channel, or by letting it disperse freely. The resulting spatial field is a set of scalar blobs and voids, with well defined widths distributions. The corresponding power spectra are not pure power laws, and their shape is evolving in time. We will present a novel method for constructing the shape of the scalar power spectra solely based on the information contained in the size distributions of the contiguous regions scalar (the blobs) and the interlacing free space (the voids).

Sunday, November 21, 2010 8:00AM - 10:10AM — Session AB Turbulent Boundary Layers I — Long Beach Convention Center 101B

8:00AM AB.00001 Extrapolating Channel Flow Data to High Reynolds Number Conditions, RICARDO VINUESA, KRIS DRESSLER, HASSAN NAGIB, IIT — Channel and pipe flows require relatively larger Reynolds numbers (Re) than boundary layers to achieve "high-Re" conditions for wall-bounded flows. Unlike the pipe case, where the log-pipe results far exceed such conditions, even the highest Re channel experiments have not reached such a state. Because of the extremely large channel facility required to achieve these high-Re conditions with good spatial resolution for the measurements, DNS may present the best hope. Available experimental and DNS results, and new data from a channel with variable aspect ratio, are used to estimate the high-Re von Kármán coefficient and the Re required to achieve it. Wall shear stress measurements with oil film interferometry were obtained over the range 7,500 < Re < 30,000 in channel flows for aspect ratios varying from 1.8 to 48. The results show that the relationship between Reynolds number and skin-friction coefficient depends on the channel aspect ratio well beyond the traditional values of 8 to 12 generally believed to ensure two-dimensionality. The results indicate that an increase by a factor of about four or five in the current Re capabilities of DNS is required to approach the asymptotic von Kármán coefficient for the channel, which likely is even lower than our recent estimates of 0.37.

8:13AM AB.00002 Modeling near-wall turbulent flows, IVAN MARUSIC, ROMAIN MATHIS, NICHOLAS HUTCHINS, University of Melbourne — The near-wall region of turbulent boundary layers is a crucial region for turbulence production, but it is also a region that becomes increasing difficult to access and make measurements in as the Reynolds number becomes very high. Consequently, it is desirable to model the turbulence in this region. Recent studies have shown that the classical description, with inner (wall) scaling alone, is insufficient to explain the behaviour of the streamwise turbulence intensities with increasing Reynolds number. Here we will review our recent near-wall model (Marusic et al., Science 329, 2010), where the near-wall turbulence is predicted given information from only the large-scale signature at a single measurement point in the logarithmic layer, considerably far from the wall. The model is consistent with the Townsend attached eddy hypothesis in that the large-scale structures associated with the log-region are felt all the way down to the wall, but also includes a non-linear amplitude modulation effect of the large structures on the near-wall turbulence. Detailed predicted spectra across the entire near-wall region will be presented, together with other higher order statistics over a large range of Reynolds numbers varying from laboratory to atmospheric flows.
8:26AM AB.00003 Coherent vorticity extraction in turbulent boundary layers using orthogonal wavelets$^1$, GEORGE KHUJADZE, Group of Fluid Dynamics, Mechanical Engineering, TU Darmstadt, Germany, ROMAIN NGUYEN VAN YEN, LMD-CNRS, Ecole Normale Superieure Paris, France, KAI SCHNEIDER, MOP2-CNRS & CMI Aix-Marseille University, France, MARTIN OBERLACK, Group of Fluid Dynamics, Mechanical Engineering, TU Darmstadt, Germany, MARIE FARGE, LMD-CNRS, Ecole Normale Superieure Paris, France — High resolution direct numerical simulation data of turbulent boundary layers are analyzed by means of wavelets. The developed anisotropic wavelet transform reinterpolates the data in the wall normal direction, originally given on a Chebyshev grid, onto an adapted dyadic grid. The contracted wavelet bases accounts for the anisotropy of the flow by using different scales in the wall normal direction and in the planes parallel to the wall. Therewith the vorticity field is decomposed into coherent and incoherent contributions. It is shown that few wavelet coefficients retain the coherent structures of the flow, while the majority of the coefficients corresponds to a structureless noise like background flow. Scale and direction dependent statistics in wavelet space quantify the properties of the total, coherent and incoherent flows as a function of the wall distance.

$^1$Part of this work has been performed during the CTR 2010 summer program, and we acknowledge partial funding by Stanford University - NASA Ames, USA.

8:39AM AB.00004 Large-eddy and direct simulations of accelerating boundary layers, JUNLIN YUAN, VALERIO GRAZIOSO, UGO PIOMELLI, Queens University, Kingston (Ontario), Canada — Turbulent boundary layers subject to a favorable pressure gradient (which induces freestream acceleration) are found in many engineering applications, such as airfoils or curved ducts. If the acceleration is sufficiently large, turbulence production decreases, and the flow reverts to a laminar or quasi-laminar state. Once the cause of relaminarization is removed, the flow retransitions to turbulence in a process that may depend critically on the residual levels of turbulent fluctuation during the relaminarization. We performed direct and large-eddy simulations (DNS and LES) of accelerating boundary layers, on smooth and rough flat plates. The DNS allows to study both the relaminarization and re-transition without requiring any turbulence model that may alter the physics. It also validates the LES, which can be extended to higher Reynolds numbers. The roughness is included using an Immersed Boundary Method. The entrainment of the rotational freestream fluid into the boundary layer plays a critical role in the formation of a well-mixed outer layer and the stabilization of the inner layer. The wall-normal and shear components of the Reynolds stresses decay more rapidly than the streamwise one, leading to a state of inactive turbulence that is advected from the upstream boundary layer into the relaminarization region. Roughness effects are limited to the near wall, but are nonetheless visible.

8:52AM AB.00005 Direct Numerical Simulation of Zero-Pressure Gradient and Sink Flow Turbulent Boundary Layers, O. RAMESH, SAURABH PATWARDHAN, Indian Institute of Science — Direct Numerical Simulations have been performed for the zero pressure gradient (ZPG) ($600 < Re_θ < 900$) and for the sink flow turbulent boundary layers ($K = 7.71 \times 10^{-7}$). A finite difference code on Cartesian grid was used to perform the simulations. Inflow generation method developed by Lund et al. was used to generate inflow boundary condition for the ZPG case. This method was slightly modified for the sink flow in view of self-similarity it possesses in the inner co-ordinates. Hence, there was no need to use empirical relations for the calculation of inlet $θ$ or $δ$ and rescaling in outer co-ordinates. The average statistics obtained from the simulations are in close agreement with the experimental as well as DNS data available in the literature. The intermittency distribution in the case of sink flow approaches zero inside the boundary layer ($y = 0.8δ$), an observation which is also confirmed by the experiments. This effect could be due to the acceleration near the boundary layer edge which suppresses the turbulent fluctuations near the boundary layer edge.

9:05AM AB.00006 Velocity-vorticity correlation structure in turbulent channel flow, JUN CHEN, JIE PEI, ZHEN-SU SHE, Peking Univ., FAZLE HUSSAIN, Univ. of Houston — A statistical structure – velocity-vorticity correlation structure (VVCS) – is defined by the amplitude distribution of a tensor field of correlation coefficients. Applied to turbulent channel flow DNS database (at $Re_θ = 180$), it captures most relevant features – qualitative and quantitative – of coherent structures near the wall, including streaks (Kline et al. 1967, JFM), inclined streamwise vortices (Jeong et al. 1997, JFM), and transverse vorticity (Jimenez & Moin 1991, JFM), etc. Associated with the streamwise velocity component (particularly $⟨uω⟩$), VVCS reveals a change of topology with increasing $y^+$, providing a physical interpretation of multiple layers of wall-bounded turbulence. The statistical structure of $⟨uω⟩$ depends on the $y^+$-location of $u$ detection. When $y^+$ is near the wall, the structure resembles streamwise vortices. But when $y^+$ is close to the center, it becomes a blob-like structure, quite different from streamwise vortices in the near-wall region. We propose that the statistical structure is adequate in modeling of the mean flow field. This study raises some doubt about unique structures in turbulent flows: consideration of a set of statistical structures is unavoidable.

9:18AM AB.00007 DNS of turbulent boundary layer over a flat plate at $Re_θ = 5200$, ANTONINO FERRANTE, KEEGAN WEBSTER, University of Washington, Seattle — We performed direct numerical simulations (DNS) of a spatially developing turbulent boundary layer over a flat plate at $Re_θ = 5200$. At this Reynolds number, our DNS results show that the overlap region of inner and outer layers extends for about 150 wall units. The turbulent inflow conditions were generated using the method of Ferrante & Elghobashi [J. Comput. Phys. 198 (2004)]. The computational domain of the main simulation is a parallelepiped with $θ = 5200$ $x$-location of $r$ $y$-value. The turbulence statistics were collected over a period of about $80$ large-eddy turnover times. These simulations were made possible thanks to our development of an optimized and scalable 3D Poisson solver, which reduced the time to integrate the incompressible Navier-Stokes equations by 40%. Our DNS results are in excellent agreement with the experimental data of DeGraaf and Eaton [J. Fluid Mech. 422 (2000)] at the same $Re_θ$.

$^1$We acknowledge NSF Advanced Support for TeraGrid Applications (ASTA) and NSF TRAC (CTS100024) Award 2010 for computing time.

9:31AM AB.00008 Direct Numerical Simulation of a Quasilaminarized Boundary Layer, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, JUAN GUILLERMO ARAYA, Swansea University, RAUL BAYOAN CAL, Portland State University — Direct Numerical Simulations of spatially-evolving turbulent boundary layers with strong favorable pressure gradients are performed. The driven force behind this investigation is elucidate the mechanisms responsible for the quasi-laminarization of the boundary layer. Budgets of the turbulent kinetic energy and the shear Reynolds stresses provide insight into the terms responsible for this phenomenon. The results also confirm the similarity analysis framework as develop by Cal and Castillo$^2$ including the redistribution of the Reynolds stresses, a significant reduction in skin friction and a pressure parameter value which falls in the quasilaminar quadrant. The prescription of stronger favorable pressure gradients is mainly manifested by a significant decrease of the production of the shear Reynolds stresses and attenuation of the velocity-pressure gradient correlation term. The latter evidence confirms the important role of pressure fluctuations on the energy exchange and transport phenomena of flow parameters.

numerical solutions that are grid independent and are not influenced by numerical errors. The efficacy of explicit-filtering to obtain a grid-independent solution is crucial due to the sensitivity of the combustion process to the extent of this mixing. However, like other subfilter quantities, variance prediction is subject to significant numerical as well as modeling errors. The criterion is designed to prevent the fields from being contaminated by the accumulated energy in the small resolved scales caused by the lack of automatic activation of the carefully designed filter is made by using the comparison between theoretically derived criterion and the same numerically calculated quantity. The criterion is developed to quantify flow structure geometry including the averaged inclination and sweep angles of both classes of flow structure at up to seven scales. These range from the half-height of the channel to several viscous length scales. Here, the inclination angle is defined on the plane of the stream-wise and wall-normal directions, and the sweep angle on the plane of the stream-wise and span-wise directions. Results for turbulent channel flow include the geometry of candidate hairpin vortices and other structures in the near-wall region, the structural evolution of near-wall vortices, and evidence for the existence and geometry of hairpin packets based on statistical inter-scale correlations.

Supported by NASA under Cooperative Agreement NNX08AD4A.

Sunday, November 21, 2010 8:00AM - 9:57AM –
Session AC Turbulence Modeling I Long Beach Convention Center 102A

8:00AM AC.00001 Development of a subgrid-scale similarity model for LES based on the physics of interscale energy transfer in turbulence, BRIAN W. ANDERSON, JULIANA A. DOMARADZKI, University of Southern California — It is well known that scale-similarity models suffer from an insufficient SGS dissipation which causes them to fail in actual, time evolving LES of turbulent flows. We determine that this failure is due to the representation of the interscale energy transfer among the resolved scales which is incorrect for purposes of LES. Alternatives to previous forms of similarity models have been evaluated with the intent of improving upon the deficiencies of these models in predicting subgrid-scale energy dissipation. Expressions describing interscale turbulence interactions have been constructed using test-filtered velocity fields and these terms have been used to formulate an alternative model that offers improvements on the predictions of global energy flux from resolved to subgrid scales. Results for wall-bounded flow simulations indicate that improved predictions of mean and RMS flow quantities are possible.

8:13AM AC.00002 Truncated Navier-Stokes Equations with the Automatic Filtering, TAWAN TANTIKUL, JULIAN DOMARADZKI — Truncated Navier-Stokes (TNS) methodology is a LES technique which does not use explicit SGS models but utilizes the periodic filtering of a solution to provide the necessary dissipation. It has been successfully validated for many turbulent flow problems. One drawback of TNS is that the filtering time interval which dictates the activation of the filtering operation in the simulation has to be prescribed in advance and is obtained by trial and error. The modified TNS procedure where the filtering interval is determined automatically during the simulations is presented. The decision for the automatic activation of the carefully designed filter is made by using the comparison between theoretically derived criterion and the same numerically calculated quantity. The criterion is designed to prevent the fields from being contaminated by the accumulated energy in the small resolved scales caused by the lack of subgrid scales. The procedure is tested in a sequence of TNS simulations for turbulent channel flow and Reynolds numbers based on the friction velocity and channel-half width up to 2000 for which detailed DNS data are available for comparison. The simulations demonstrate a convergence toward their respective DNS results once the near wall structures are resolved. The results at high grid resolution exhibit the independence of the quality of the results on the filtering criterion at a certain constant filtering interval.

8:26AM AC.00003 Analysis of numerical errors in subfilter scalar variance models for large eddy simulation, COLIN R. HEYE, COLLEEN M. KAUL, VENKAT RAMAN, The University of Texas at Austin — In conserved scalar methods for large eddy simulation (LES) of combustion, the subfilter scalar variance characterizes the degree of small scale mixing. Accurate scalar variance modeling is crucial due to the sensitivity of the combustion process to the extent of this mixing. However, like other subfilter quantities, variance prediction is subject to significant numerical as well as modeling errors. Here, we compare algebraic and transport equation-based variance models using a coupled DNS-LES a posteriori analysis approach. Model performance is evaluated for the cases of homogeneous isotropic turbulence (HIT) and turbulent jet flow, which provide complementary information. In the HIT case, results using a variety of finite difference schemes are compared to spectrally computed LES and DNS results. Algebraic models are found to incur large numerical errors while scalar dissipation rate modeling is the foremost source of error for transport equation models. The jet flow case extends the comparison to a more realistic configuration and allows additional numerical and physical factors to be considered, such as variable stencil numerical schemes and persistent large scale gradients.

8:39AM AC.00004 Grid-independent large-eddy simulation of compressible turbulent flows using explicit filtering, DONGHYOUNG YOU, Carnegie Mellon University, SANJEEB BOSE, PARVIZ MOIN, Center for Turbulence Research — One of the most notable drawbacks associated with the conventional implicit-filter LES is that the simulation result is dependent on the numerical grid employed due to the inherent dependence of the filtering operation on the numerical discretization. As a consequence of the grid-dependency, the implicit-filter LES is sensitive to numerical errors. In the present study, the use of explicit filtering in LES of compressible turbulent flows, is investigated in order to obtain numerical solutions that are grid independent and are not influenced by numerical errors. The efficacy of explicit-filtering to obtain a grid-independent solution of incompressible turbulent channel flow has been successfully demonstrated in the previous research (Bose, Moin & You, Phys. Fluids, 2010). In the present study, an effective methodology for explicit-filter LES is developed and validated for compressible turbulent flows. The convergence of simulations using a fixed filter width with varying mesh resolutions to a true LES solution will be analyzed, with particular attention to the performance of the chosen subgrid-scale model. Results from explicit-filtering LES of compressible turbulent flow through a channel with periodic contractions will be presented.

Supported by the Center for Turbulence Research Summer Program
9:05AM AC.00006 A mixed LES model based on the residual-based variational multiscale formulation of compressible flows, JIANFENG LIU, ZHEN WANG, ASSAD OBERAI, Department of Mechanical, Aerospace and Nuclear Engineering, Rensselaer Polytechnic Institute — In the residual-based variational multiscale (RBVM) formulation of large eddy simulation (LES) a projection operator is used to separate the solution of the Navier-Stokes equations into coarse and fine scales. The coarse scale equations are solved numerically while the fine scale equations are solved analytically. In particular, an algebraic approximation for the fine scale velocities is derived wherein they are expressed in terms of the residual of the Navier-Stokes equations applied to the coarse scale solution. We have recently demonstrated that while the RBVM model accurately represents the subgrid scale fluctuation, it under-estimates the contribution from the Reynolds-stress term. To remedy this we add to it a Smagorinsky eddy viscosity which provides a good approximation to the Reynolds-stress term. This leads us to a mixed model capable of accurately modeling all components of the subgrid stress. In this talk we extend this model to compressible flows and use it to predict the decay of compressible homogeneous isotropic turbulence. We note that the mixed model yields better agreement with direct numerical simulation than either of its components: the dynamic Smagorinsky model and the residual-based VMS model.

9:18AM AC.00007 Evaluation of Leray and Verstappen regularizations in LES, without and with added SGS modeling, G. WINKELMANS, N. BOURGEAIS, Y. COLLET, M. DUPONCHEEL, Universite catholique de Louvain (UCL) — Institute of Mechanics, Materials and Civil Engineering (iMMC) — Regularization approaches (Leray and Verstappen) for the “restriction in the rate of production of small-scales” in turbulence simulations have regained some interest in the LES community. Their potential is here investigated using the best numerics (dealiased pseudo-spectral code) and on two cases: transition of the Taylor-Green vortex (TGV) and its ensuing turbulence, decaying homogeneous isotropic turbulence (HIT). The filtered velocity fields are obtained using discrete filters, also of various orders. Diagnostics include energy, enstrophy and spectra. The performance of the regularizations is first evaluated on the TGV in inviscid mode (Re=1); then in viscous mode: 256³ DNS at Re = 1600, 128³ LES at Re = 5000 (compared to 1024³ DNS). Although they indeed delay the rate of production of small scales, they cannot sustain LES when the flow has become turbulent: the small scales are still too energized. Added subgrid-scale (SGS) modeling is thus required. The combination of regularization and SGS modeling (here using the RVM multiscale model) is then also evaluated. Finally, 128³ LES of fully developed HIT at very high Re is also investigated, providing the asymptotic behavior. The regularizations help increase the true inertial subrange obtained with the RVM model.

9:31AM AC.00008 Curve fitting 3-D experimental turbulent flows with the poor man’s Navier-Stokes equation, J. M. MCDONOUGH, University of Kentucky, T. C. MITCHELL, P. L. Dunbar High School, S. C. C. BAILLY, University of Kentucky — The 3-D poor man’s Navier-Stokes (PMNS) equation is a discrete dynamical system (DDS) whose solutions retain much of the dynamical behavior of the partial differential equations from which it is derived, and yet is very easily executed—far faster than real time. We briefly outline derivation of this DDS and then discuss a general procedure for curve fitting DDSs to chaotic experimental data. This technique (first introduced by McDonough et al., Appl. Comp. Math. 1998 and later used by Yang et al., AIAA J. 2003 in a 2-D Navier–Stokes setting) employs a least-squares method to generate a global (long-time) fit of chaotic data that produces details of experimental time series in a manner more appropriate for representing fluid turbulence (including sensitivity to initial conditions) than often used short-time extrapolation techniques can. We apply this least-squares approach to three-component velocity measurements in grid turbulence described by Bailey and Tavoularis, J. Fluid Mech. 2008, and demonstrate that the PMNS equation can reproduce the structure of all three experimental velocity components. We present comparisons of time series, energy spectra, and other typical turbulence statistics, e.g., flatness and skewness. A possible application of such curve fits would be to real-time control of physical turbulent fluid flows.

9:44AM AC.00009 Large-eddy simulation of the upper ocean mixed layer, ANDRES TEJADA-MARTINEZ, University of South Florida — Large-eddy simulations (LES) of the wind-driven upper ocean mixed layer with and without wave-current interaction are presented. Wave-current interaction is parameterized through the well-known Craik-Leibovich (C-L) vortex force appearing in the momentum equation and generating Langmuir circulation (LC). LC consists of pairs of parallel counter-rotating vortices aligned in the direction of the wind characterizing the turbulence (i.e. the Langmuir turbulence) advected by the mean flow. LES grid-scale (SGS) closure is given by a traditional Smagorinsky eddy viscosity model for which the model coefficient is derived following similarity theory in the near-surface region. Alternatively, LES closure is given by the dynamic Smagorinsky model (DSM) for which the model coefficient is computed dynamically as a function of the flow. The validity of the DSM for parameterizing the viscous sublayer is assessed and a modification to the surface stress boundary condition based on log-layer behavior is introduced improving the performance of the DSM. Furthermore, in the simulation with wave-current interaction, the implicit LES grid filter leads to LC grid-scales requiring explicit spatial filtering of the C-L vortex force in place of a suitable SGS parameterization.

1 This work was supported through NSF grant ANT-0838988.

Sunday, November 21, 2010 8:00AM - 10:10AM — Session AD CFD: Algorithms I — Long Beach Convention Center 102B

8:00AM AD.00001 Incompact3d: a powerful tool to tackle turbulence problems with up to hundreds of thousands computational cores, SYLVAIN LAIZET, Imperial College London, NING LI, NAG — Understanding the nature of complex turbulent flows remains one of the most challenging problems in classical physics. Significant progress has been made recently using High Performance Computing, and Computational Fluid Dynamics is now a credible alternative to experiments and theories in order to understand the rich physics of turbulence. In this work, we present an efficient numerical tool called “Incompact3d” that can be coupled with massive parallel platforms in order to simulate turbulence problems with such complexity as possible, using up to hundreds of thousands computational cores by means of Direct Numerical Simulation. “Incompact3d,” that solved the incompressible Navier-Stokes equation, is a finite-difference code (sixth order schemes in space) that can be combined with an Immersed Boundary Method (IBM) in order to simulate flow with complex geometry. The originality of this code is that the Poisson equation is solve in the spectral space in the framework of the modified wave number. We will demonstrate that “Incompact3d” is a powerful tool that can undertake DNS with up to hundreds of thousands computational cores thanks to an efficient 2D domain decomposition.
and the semi-implicit scheme will be compared with several numerical problems. Moreover, the computations of the velocity and pressure are decoupled. The proposed scheme allows the use of time step sizes considerably larger than the commonly used semi-implicit type schemes. The scheme is based on a velocity correction type formulation, such as projection or fractional step methods, which can affect stability and accuracy of numerical schemes. It is well known that the accuracy of numerical methods degrades rapidly with increase in mesh refinement. Near wall boundaries, spatial derivatives are specified to be zero at the outset and never relying on continuum relations afterward. This ensures uniform high order of accuracy in time for all fields, including pressure. The semi-implicit method of fractional step methods, has limited temporal accuracy as a result of matrix splitting errors, or introduce errors near the domain boundaries, resulting in weakly convergent solutions. We recast the Navier-Stokes incompressibility constraint as a pressure Poisson equation with velocity dependent boundary conditions. Applying the remaining velocity boundary conditions to the momentum equation, we obtain a pair of equations, for the primary variables velocity and pressure, equivalent to the incompressible Navier-Stokes. Since in this recast system the pressure can be efficiently recovered from the velocity, this reformulation is ideal for numerical marching methods. The equations can be discretized using a variety of methods, in principle to any desired order of accuracy. In this work we illustrate the approach with a 2-D second order finite difference scheme on a Cartesian grid, and devise an algorithm to solve the equations on grids with curved (non-conforming) boundaries, including a case with a non-trivial topology (a circular obstruction inside the domain). This algorithm achieves second order accuracy in the $L^2$ norm for both the velocity and the pressure. The scheme has a natural extension to 3-D.

An efficient method for the incompressible Navier-Stokes equations on irregular meshes with no-slip boundary conditions, high order up to the boundary. RUBEN ROSALES, Department of Mathematics, MIT — Common efficient schemes for the incompressible Navier-Stokes equations, such as projection or fractional step methods, have limited temporal accuracy as a result of matrix splitting errors, or introduce errors near the domain boundaries, resulting in weakly convergent solutions. We recast the Navier-Stokes incompressibility constraint as a pressure Poisson equation with velocity dependent boundary conditions. Applying the remaining velocity boundary conditions to the momentum equation, we obtain a pair of equations, for the primary variables velocity and pressure, equivalent to the incompressible Navier-Stokes. Since in this recast system the pressure can be efficiently recovered from the velocity, this reformulation is ideal for numerical marching methods. The equations can be discretized using a variety of methods, in principle to any desired order of accuracy. In this work we illustrate the approach with a 2-D second order finite difference scheme on a Cartesian grid, and devise an algorithm to solve the equations on grids with curved (non-conforming) boundaries, including a case with a non-trivial topology (a circular obstruction inside the domain). This algorithm achieves second order accuracy in the $L^2$ norm for both the velocity and the pressure. The scheme has a natural extension to 3-D.

A Finite-Volume ADI Method for Simulation of Incompressible Flows on Curvilinear Grids. SATBIR SINGH, DONGHYUN YOU, Carnegie Mellon University — A second-order accurate finite-volume-based alternating direction implicit (ADI) method is proposed for the solution of incompressible Navier-Stokes equations on structured curvilinear meshes. Numerical accuracy and stability at high Reynolds numbers are achieved with the selection of the discrete operators and solution algorithms which assure discrete kinetic energy conservation in the inviscid limit. Unlike the conventional finite-difference-based ADI schemes, in which the factorization is performed along the transformed generalized-coordinate directions, in the proposed method, the discretized equations are factored along the curvilinear mesh lines without coordinate transformation. The accuracy, stability, and efficiency of the proposed method are assessed in simulations of an unsteady convection-diffusion equation on Cartesian and skewed meshes, and simulations of lid-driven cavity flow, flow over a circular cylinder, and turbulent channel flow. In the proposed method, the computational cost required for the solution of momentum equations is found to be 3 to 5 times smaller than that required when a bi-conjugate gradient stabilized (BCGSTAB) iterative method is employed.

A Finite-Volume ADI Method for Simulation of Incompressible Flows on Curvilinear Grids, SATBIR SINGH, DONGHYUN YOU, Carnegie Mellon University — A second-order accurate finite-volume-based alternating direction implicit (ADI) method is proposed for the solution of incompressible Navier-Stokes equations on structured curvilinear meshes. Numerical accuracy and stability at high Reynolds numbers are achieved with the selection of the discrete operators and solution algorithms which assure discrete kinetic energy conservation in the inviscid limit. Unlike the conventional finite-difference-based ADI schemes, in which the factorization is performed along the transformed generalized-coordinate directions, in the proposed method, the discretized equations are factored along the curvilinear mesh lines without coordinate transformation. The accuracy, stability, and efficiency of the proposed method are assessed in simulations of an unsteady convection-diffusion equation on Cartesian and skewed meshes, and simulations of lid-driven cavity flow, flow over a circular cylinder, and turbulent channel flow. In the proposed method, the computational cost required for the solution of momentum equations is found to be 3 to 5 times smaller than that required when a bi-conjugate gradient stabilized (BCGSTAB) iterative method is employed.

Discretely conservative, non-dissipative, and stable collocated method for solving the incompressible Navier-Stokes equations. REETESH RANJAN, CARLOS PANTANO, University of Illinois at Urbana-Champaign — We present a new method for solving the incompressible Navier-Stokes equations. The method utilizes a collocated arrangement of all variables in space. It uses centered second-order accurate finite-difference approximations for all spatial derivatives and a third-order IMEX approach for time integration. The proposed method ensures discrete conservation of mass and momentum by discretizing the conservative form of the equations from the outset and never relying on continuum relations afterward. This ensures uniform high order of accuracy in time for all fields, including pressure. The pressure-momentum coupled equations can be easily segregated and solved sequentially, as in the pressure projection method but without a splitting error. In this approach there are no spurious kernel modes, checkerboard, in the embedded elliptic pressure problem. The method has been applied to different canonical problems, including a fully periodic box, a periodic channel, an inflow-outflow channel and a lid-driven cavity flow. Near wall boundaries, spatial derivatives are obtained using the weak form of the conservation equations, similar to a finite element approach. The results from some of the sample cases will be presented to illustrate the features of the method.

Robust and accurate finite volume method on highly skewed unstructured meshes, HYUNCHUL JANG, KRISHNAN MAHESH, Aerospace Engineering, University of Minnesota — Geometric complexity often causes highly skewed meshes, which can affect stability and accuracy of numerical scheme. It is well known that the accuracy of numerical methods degrades rapidly with increase of internal angles in skewed elements. A regularized least squared method with multi-dimensional slope limiters is derived for convective flux reconstruction. Two deferred correction methods are also derived for diffusive flux reconstruction and the Poisson equation. Those methods show considerable improvement and converge even on highly skewed meshes. Also, the second-order accuracy is held with those methods on both of mildly and highly skewed meshes. This numerical method is applied to a realistic complex problem such as the large eddy simulation for marine propulsor in an extreme operating condition. This work is supported by the United States Office of Naval Research under ONR Grant N00014-02-1-0978.

Large Eddy Simulation of Flow Over Surface-Mounted Cube Using a Spectral Element Method. SRIHARSHA KANDALA, DIETMAR REMPFFER, IIT, Chicago — Unsteady three dimensional flow over a surface-mounted cube, with its rich set of features like flow turbulence, upstream boundary layer separation, curved mixing layer, unsteady three dimensional wake, etc., provides a test case for evaluating the performance of CFD codes. We are developing a parallel spectral element code, SpecSolve, with the objective of modeling incompressible flows in complex geometries. The code is based on the fractional step method and uses the operator-integrating factor splitting scheme for temporal integration. In this talk, we provide a brief overview of the algorithm and implementation details. We present results from large-eddy simulations of flow over a surface-mounted cube using SpecSolve. The Reynolds number, based on bulk flow velocity and height of the cube, is 40,000. The dynamic Smagorinsky model is used for modeling turbulence. These results are compared with experimental data of Martinuzzi and Tropea, LES results of Shah and Ferziger and our FLUENT LES simulations.

A time-stepping scheme for flow simulations that allows the use of large time step sizes. STEVEN DONG, Purdue University — We present a time-stepping scheme for incompressible Navier-Stokes equations that allows the use of time step sizes considerably larger than the commonly used semi-implicit type schemes. The scheme is based on a velocity correction type formulation, and involves only linear algebraic equations after discretization. Moreover, the computations of the velocity and pressure are decoupled. The proposed scheme and the semi-implicit scheme will be compared with several numerical problems.

\textsuperscript{3}This work is partially supported by NSF.
spectra. receptivity to free-stream vorticity and determine the relevance of these mechanisms in boundary layers exposed to turbulent free streams with different frequency wavenumbers and frequencies. The associated receptivity mechanisms are characterized in terms of receptivity coefficients. We discuss linear and nonlinear parametric study of receptivity to localized roughness elements with various length scales, shapes and downstream positions and to vortical modes with different disturbances. These sources trigger steady or traveling Görtler modes composed of counter-rotating streamwise rolls and streamwise streaks. We present a of curvature is considered using direct numerical simulations. The boundary layer forming on the plate is exposed to wall roughness and to free-stream vortical \[ \text{Re} \rightarrow \infty \]. Motivated by results in the mathematical literature on the “Physically Reasonable” solutions (Finn & Smith, 1967), we ensure our solutions are characterized by a suitable rate of decay at infinity. Since this cannot be achieved in classical CFD methods based on a truncation of the infinite domain to a finite “computational box”, we propose an alternative approach in which the Navier–Stokes equation is rewritten as a perturbation to the Oseen equations whose solutions are determined in a semi-analytic framework. The resulting problem is discretized using a combination of Fourier–Galerkin and tau–collocation method based on the rational Chebyshev functions. We will present results showing how the wake structure changes with increasing Reynolds number.

9:57AM AD.00010 Variational integrator preserving Lie-symmetry , MARX CHHAY, Univ. La Rochelle, France — Many physical systems can be expressed with a Lagrangian formalism. The underlying role of the variational symmetries occurring in the computation of the dynamics equation reveals the intrinsic conservation properties of the system. For numerical design, it is well-known that the discrete version of the variational derivation of finite dimensional time independent Lagrangian systems yields a symplectic integrator that preserves exactly the discrete energy when the time step is considered as a variable. But it is not enough for the integrator to preserve the other conservation laws. Indeed the discrete Lagrangian must also be invariant under the variational symmetries. Such Lie-symmetry variational integrators can be constructed thanks to the concept of moving frames. Numerical properties are shown on academic examples.

Sunday, November 21, 2010 8:00AM - 10:10AM – Session AE Instability: Boundary Layers I Long Beach Convention Center 102C

8:00AM AE.00001 A Comparative Study of Subgrid Scale Models, for Prediction of Transition in Turbulent Boundary Layers\(^1\) , TARANEH SAYADI, PARVIZ MOIN, CTR at Stanford University — Large eddy simulation of subharmonic transition of a spatially developing zero pressure gradient boundary layer at \(Ma = 0.2\) is investigated using three different subgrid scale (SGS) models: Dynamic Smagorinsky [1], dynamic model involving the SGS kinetic energy [2] and dynamic scale similarity model. The interest lies in assessing the capability of each model in predicting the location of transition and the overshoot in the skin friction coefficient which is specific to this transition scenario. In the case of dynamic Smagorinsky model results were obtained for four different grid resolutions and it is observed that the location of transition is largely unaffected, indicating robust performance of the dynamic model in this respect. However, after breakdown and in the turbulent region the simulations with coarsest grids produce insufficient eddy viscosity to sustain the correct value of skin friction along the plate. As a result the coarsest resolution is employed to compare the performance of these three models. The point of transition is estimated correctly in each case, but the value of the overshoot and the turbulent statistics are affected by the model. [1] Moin P. et. al. Phys Fluids A, 3(11), 2746-2757, 1991. [2] Ghosal. S. et. al. JFM, 286, 229- 255, 1995.

\(^1\)Supported by the PSAAP program of DoE

8:13AM AE.00002 Sensitivity of non-modal instabilities to base-flow modifications , LUCA BRANDT, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden, JAN PRALITS, DIMEC, University of Salerno, Italy, DENIS SIPP, OLIVIER MARQUET, ONERA/DAFE, Meudon, France — Non-modal analysis determines the potential for energy amplification in stable flows. This is quantified in the frequency domain by the singular values of the resolvent operator and in the time domain by the singular values of the evolution operator. The present work extends previous analysis on the effect of base flow variation on flow stability by considering the sensitivity of the flow non-modal behavior. Using a variational technique, we derive an analytical expression for the gradient of the resolvent norm of the system with respect to a base flow modification and show how it depends on the optimal forcing and optimal response. The potential of such an approach is illustrated for zero-pressure-gradient boundary layers where the different instability mechanisms of wall-bounded shear flows are all at work. Results confirm previous findings and clearly indicate that base flow modifications can stabilize Tollmien-Schlichting waves whereas the amplification of streamwise streaks is more difficult to hamper. This result is now explained simply examining the expression for the gradient of the resolvent norm.

8:26AM AE.00003 Formation of roll/streaks structures in boundary layers as an instability of turbulence/mean-flow interaction , PETROS IOANNOU, University of Athens — Stochastic Structural Stability Theory (SSST) provides a deterministic nonlinear dynamical system for evolving the statistical mean state of a turbulent system. In this presentation SSST is applied to the problem of understanding the origin of the roll/streak structures that arise from free stream turbulence and are associated with bypass transition in boundary layers. Roll structures in the cross-stream/spanwise plane and associated streamwise streaks are shown to arise as a linear instability of interaction between the free stream turbulence and the mean flow. In this interaction Reynolds stresses arising from free stream turbulence are organized by perturbation streamwise streaks to force perturbation rolls giving rise to an amplification of the streamwise streak that in turn further organize the free stream turbulence to produce through this feedback interaction an instability of the roll/streak/turbulence complex. The dominant turbulent perturbation structures involved in supporting the roll/streak/turbulence complex instability are the non-normal oblique optimal perturbations. The emergence of the roll/streak structure arises at a bifurcation in the parameter of free stream turbulence intensity. The instability eventually equilibrates nonlinearly producing perturbation stable streaks and vortex circulations in agreement with the observed structures in transitional boundary layers.

8:39AM AE.00004 Receptivity, Growth and Breakdown of Görtler Vortices , TAMER ZAKI, Imperial College London, UK, LARS-UVE SCHRADER, LUCA BRANDT, KTH Mechanics, Stockholm, Sweden — The flow over a concave plate with constant radius of curvature is considered using direct numerical simulations. The boundary layer forming on the plate is exposed to wall roughness and to free-stream vortical disturbances. These sources trigger steady or traveling Görtler modes composed of counter-rotating streamwise rolls and streamwise streaks. We present a parametric study of receptivity to localized roughness elements with various length scales, shapes and downstream positions and to vortical modes with different wavenumbers and frequencies. The associated receptivity mechanisms are characterized in terms of receptivity coefficients. We discuss linear and nonlinear receptivity to free-stream vorticity and determine the relevance of these mechanisms in boundary layers exposed to turbulent free streams with different frequency spectra.
8:52AM AE.00005 Secondary Instability of Roughness-Induced Transient Growth1, NICHOLAS DENISSEN, EDWARD WHITE, Texas A&M University — Optimal perturbation methods have provided the primary means of studying transient growth in theoretical and computational frameworks. Many interesting aspects of transient growth have been analyzed with these approaches, including the onset of secondary instability leading to transition-to-turbulence. However, while optimal perturbations are those that experience the most transient energy growth, this is not synonymous with the perturbations most likely to cause transition. The present work performs stability analysis of the flow field behind an array of periodic roughness elements. The time-dependent, pure-diffusion base state. This problem has been addressed in several previous studies using a variety of approximations. We present a simple making the brine slightly denser than “pure” brine. Motivated by this, we consider conditions for free convection in a porous medium from a one-dimensional, steady boundary-layer flow. Three types of spanwise-periodic discrete roughness elements are used. The influence of wall oscillation on the boundary-layer thicknesses, are calculated, showing that a critical liquid thickness can be found under which no instability can occur. The latter point coincides approximately with previous work on this problem will also be provided. The authors acknowledge the support of the NSF GRFP and the AFSOR and NASA through the National Hypersonic Science Center for Laminar-Turbulent Transition, AFSOR Grant FA9550-09-1-0341.

9:05AM AE.00006 Boundary-layer receptivity of three-dimensional roughness arrays on a swept-wing 1, LAUREN HUNT, WILLIAM SARIC, Texas A&M University — This experimental study extends the knowledge base of swept-wing receptivity mechanisms to three-dimensional surface roughness arrays, quantifying the relationship between surface roughness height and initial disturbance amplitudes within a boundary layer that is dominated by a crossflow instability. The experimental configuration includes the ASU(67)-0315 swept-wing installed in the low-turbulence Klebanoff-Saric Wind Tunnel at Texas A&M University. It has a 45-degree sweep, 1.83m chord and a pressure minimum at 71% chord. Results of naphthalene flow visualization and detailed boundary-layer scans using hotwire anemometry are provided.

9:18AM AE.00007 Bypass transition delay via oscillating Stokes layers , PHILIPPI HACK, LUCA BURINI, TAMER ZAKI, Imperial College London — The breakdown of laminar boundary layers to turbulence is accompanied by a large increase in skin friction drag. Therefore, flow modification strategies are sought in order to delay transition and reduce drag. Our work addresses the influence of wall oscillation on the proceedings of bypass transition, and in particular on the amplification of boundary layer streaks. Direct numerical simulations demonstrate that appropriate choice of oscillation amplitude and frequency can yield a considerable reduction in local skin friction, and thereby drag. Linear, transient analyses are performed in order to explain the physical mechanism and the optimal parameters of wall oscillation. Finally, it is shown that an overall energetic advantage is possible, where the reduction in propulsion power outweighs the required input into the wall movement.

9:31AM AE.00008 Linear stability analysis of an evaporating binary liquid layer with fully transient reference profiles 1, HATIM MACHRAFI, Universite de Liege, ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles, PIERRE DAUBY, Universite de Liege — This study deals with an evaporating horizontal binary-liquid layer (aqueous solution of ethanol; mass fraction 0.1) in contact with air with an imposed transfer distance. Fully transient solutions for the reference state are first calculated by means of a finite difference method. Then, the linear stability problem is solved using the frozen-time approach. After decomposition into normal modes, we obtain a problem for the eigenvalues, depending on the time as a parameter, which is numerically solved using the Chebyshev decomposition. Solutal and thermal Rayleigh-Benard-Marangoni instabilities are taken into account together with the Soret effect. The critical times needed for instability to occur and corresponding liquid thicknesses, are calculated, showing that a critical liquid thickness can be found under which no instability can occur. The latter point coincides approximately with the diffusive boundary layer reaching the bottom of the liquid layer. For instance, for a gas/liquid thickness layer ratio fixed at 10, the critical liquid thickness appears to be rather small, about 18.5 µm, which is illustrative of the general tendency.

Supported by ESA & BELSPO PRODEX projects, and by FRS - FNRS.

9:44AM AE.00009 Inviscid Stability Analysis of Chemically Reacting Boundary Layers in Binary Gas Mixtures 1, JILL KLEINTZMAN, ERMAN ULKER, ANATOLI TUMIN, University of Arizona — The inviscid stability of boundary layers in binary mixtures of nitrogen and oxygen in thermal equilibrium is investigated. Both temporal and spatial frameworks are considered. With the assumption of inviscid flow, the simplicity of the governing equations allows for a clear, direct comparison between the structure of the eigenvalue spectrums of real and perfect gases. Our results indicate that qualitatively the eigenvalue spectrum of a real gas should be similar to that of a perfect gas. When we examine the breakup of real gas effects on the first and second modes, we find that real gas effects act to stabilize the first and destabilize the second. This can be attributed to the deflection of temperature and species in the equilibrium. When we evolve the real gas effect of changes in species concentrations due to chemical reactions, we find that this effect alone stabilizes the second mode, which agrees with studies of flows at finite Reynolds number. This could be the result of energy being absorbed and used for dissociation rather than for the growth of disturbances.

This work was sponsored by AFSOR/National Center for Hypersonic Research in Laminar-Turbulent Transition and by the Air Force Office of Scientific Research, USAF, under grant No. FA9550-08-1-0322.

9:57AM AE.00010 Stability of corner boundary layers , JIM DENIER, NATHANIEL JEWELL, The University of Adelaide — We reconsider the problem of the stability of the flow in an internal corner, focusing attention on determining both the most unstable mode and the critical Reynolds number for instability. Our results predict that the zeroth, the so called, inviscid mode becomes unstable first with a critical Reynolds number of 43,500. Comparison with previous work will also be provided.

Sunday, November 21, 2010 8:00AM - 9:57AM – Session AF Porous Media I: CO2 Sequestration Long Beach Convention Center 103A

8:00AM AF.00001 Onset and cessation of porous convection in the context of geological carbon sequestration , ANJA SLIM, Harvard University, T.S. RAMAKRISHNAN, Schlumberger-Doll Research, Cambridge, MA 02139 — In geological carbon sequestration, CO2 injected into a saline aquifer is less dense than the resident brine and floats above it. It is slightly soluble in brine and progressively dissolves, making the brine slightly denser than “pure” brine. Motivated by this, we consider conditions for free convection in a porous medium from a one-dimensional, time-dependent, pure-diffusion base state. This problem has been addressed in several previous studies using a variety of approximations. We present a simple but rigorous calculation, showing where in time and wavenumber space a perturbation exists (of infinitesimal or finite amplitude) whose mean square amplitude grows. The critical Rayleigh-Darcy number, Ra, below which instability cannot occur is Ra = 32.50. Above Ra ≈ 100, the earliest possible onset time becomes independent of porous-layer thickness. We discuss implications for realistic reservoir conditions.
8:13AM AF.00002 Dissolution-driven convection in porous media: Experiments, MAHESH BANDI, ANJA SLIM, L. MAHDEVAN, School of Engineering and Applied Sciences, Harvard University — The carbon geo-sequestration proposal has received extensive theoretical and numerical scrutiny in recent times, but few supporting experimental observations exist. Here we experimentally investigate this proposal, and study the onset of convection away from a time-dependent pure diffusion base state. The experimental setup comprises a Hele-Shaw cell containing Potassium Permanganate (representing carbon dioxide) and water (representing brine). Upon coming in instantaneous contact, Potassium permanganate dissolves in water and produces an initially clean diffusion layer. The Potassium Permanganate solution being slightly denser than pure water, becomes gravitationally unstable and sinks by forming fingers, and heralds the onset of convection. Inclining the Hele-Shaw cell at steep angles extends the diffusive range and allows us to capture the cross-over from diffusive to convective regimes. System dynamics are captured by digitally imaging the light transmitted across the Hele-Shaw cell by a back-lit diffuse illumination source. Pre-calibration of the transmitted light at various Potassium Permanganate concentrations allows us to convert the digital images into concentration fields and calculate the local and global flux in the system. We present preliminary results from this experimental study.

8:26AM AF.00003 The Effect of Geothermal Reaction on Convective Mixing in a Gravity-annly Unstable Diffusive Boundary Layer in Porous Media: Geological Storage of CO₂ in Saline Aquifers, KARIM GHAZMOT, HASSAN HASSANZADEH, JALAL ABEDI, University of Calgary — The storage of carbon dioxide and acid gases in deep geological formations is considered a promising option for mitigation of greenhouse gas emissions. Understanding of the primary mechanisms, such as convective mixing and geochemistry that affect the long-term geostorage process in deep saline aquifers is of prime importance. First, a linear stability analysis of an unstable diffusive boundary layer in porous media is presented, where the instability occurs due to a density difference between the carbon dioxide saturated brine and the resident brine. The linear stability results have revealed that geochemistry stabilizes the boundary layer. A detailed physical discussion is also presented with an examination of vorticity and concentration eigenfunctions and streamlines’ contours to reveal how the geochemical reaction may affect these physical terms. Second, nonlinear direct numerical simulations are presented, in which the evolution of density-driven instabilities for different reaction rates are discussed. The results indicate that the boundary layer will be more stable for systems with a higher rate of reaction. However, the quantitative analyses show that more carbon dioxide may be removed from the supercritical free phase as the flux at the boundary is higher for flow systems coupled with stronger geochemical reactions.

8:39AM AF.00004 Drainage in Two-dimensional Porous Media: From Capillary Fingering to Viscous flow, HUGUE BODIGUEL, CHRISTOPHE COTTIN, ANNIE COLIN, LOF, CNRS UMR 5258, univ. Bordeaux 1, Rhôda — We report some experimental results on two-phase flows in model 2D porous media. Standard microfluidic techniques are used to fabricate networks of straight microchannels having a controlled throat size distribution. We focus on the drainage of a wetting fluid by a non-wetting one of various viscosities and take advantage of image analysis to characterize the velocities of the menisci that are simultaneously moving. In the range of applied capillary numbers (Ca) from 10⁻⁵ to 10⁻⁷, the system exhibits a clear transition from a fractal front to a stable front, which depends mainly on the size heterogeneity of the medium. The experimental results are accounted by a simple model that accounts for the scaling behaviour of the local velocities as a function of Ca. We also obtain a very good quantitative agreement when comparing the experimental results to numerical simulations based on a pure network model. This allows us to propose a general prediction of the capillary fingering extent as a function of the capillary number, the channel geometries and the pore size heterogeneity.

8:52AM AF.00005 Leakage from inclined porous reservoirs, PAWEL ZIMOCH, JEROME NEUFELD, DOMINIC VELLA, University of Cambridge — We investigate the effect of localized leakage on the injection of buoyant fluids in porous, inclined reservoirs, with application to the geological storage of CO₂. We consider a simplified two-dimensional geometry and find that the resulting gravity current reaches a steady-state shape apart from a nose, which propagates at constant velocity. Crucially, this means that the efficiency of storage (defined as the instantaneous proportion of the injected fluid that does not leak) tends to a finite value at late times. This is in contrast to previous studies of localized leakage in horizontal reservoirs, which found that the efficiency of storage tends to zero at late times. We analyze the steady-state efficiency and relevant time scales for leakage points located upslope and downslope of the injection point using analytical and numerical methods, and compare our findings with the results of model laboratory experiments. Finally, we consider the implications of our results for the geological storage of CO₂ in the presence of sloping cap rocks compromised by the presence of fractures or fissures.

9:05AM AF.00006 Convective dissolution in porous media, JEROME NEUFELD, University of Cambridge, MARC HESSE, University of Texas at Austin, AMIR RIAZ, University of Maryland, MARK HALLWORTH, University of Cambridge, HAMDI TCHELEPI, Stanford University, HERBERT HUPPERT, University of Cambridge — Motivated by the geological storage of buoyant carbon dioxide (CO₂) we investigate dissolution of CO₂ into brine which increases security of storage over time. The rate of CO₂ dissolution is determined by convection in the brine driven by the increase of brine density with CO₂ saturation. We present a new analogue fluid system that reproduces the nonlinear density behaviour of CO₂ and brine. We show that the convective flow (proportional to the Rayleigh number) and the 4/5 power through a combination of laboratory experiments and high-resolution numerical simulations, in contrast with a classical linear relationship. This relationship allows us to extrapolate from the laboratory scale to geophysical scales. A scaling argument that incorporates the effect of the large-scale flow on mixing at the CO₂-brine interface confirms this nonlinear relationship for the convective flow and provides a physical picture of high Rayleigh number convection in a porous medium. The resultant model makes quantitative predictions of the CO₂ dissolution rates in natural and anthropogenic CO₂ accumulations. For example, at the Sleipner field we estimate a dissolution rate of roughly 10% of the annual injected mass suggesting that storage security is significantly enhanced.

9:18AM AF.00007 Spreading and dissolution of CO₂ in horizontal aquifers: theory and experiments, CHRISTOPHER MACMINN, Massachusetts Institute of Technology, JEROME NEUFELD, Cambridge University, MARC HESSE, University of Texas at Austin, HERBERT HUPPERT, Cambridge University — Injection of carbon dioxide into saline aquifers is widely regarded as a promising tool for reducing atmospheric CO₂ emissions. While an accurate assessment of the post-injection spreading and migration of the CO₂ is essential for estimates of storage security, many of the physical processes controlling CO₂ migration are poorly understood. CO₂ is buoyant relative to groundwater at reservoir conditions. This is undesirable because the presence of a pre-existing well or fracture, or the activation of a fault, could lead to leakage. It is well known, however, that the dissolution of CO₂ increases the density of the groundwater, resulting in convective currents that dramatically enhance CO₂ dissolution. Once dissolved, the CO₂ is considered to be securely stored within the subsurface. Recent numerical and experimental work has led to a greatly improved understanding of the resulting rate of CO₂ dissolution into groundwater. Here, we use analog experiments and simple theoretical models to study dissolution from a plume of CO₂ as it spreads upward against the caprock in an aquifer of finite thickness. We show that the interaction between spreading, dissolution, and the finite thickness of the aquifer has a strong influence on the ultimate distribution of the CO₂.

9:31AM AF.00008 Unstable Diffusion Layers: Laboratory Experiments on Carbon Sequestration Phenomena, ROBERT ECKE, SCOTT BACKHAUS, KONSTANTIN TURITSYN, Los Alamos National Laboratory — The sequestration of carbon dioxide in aqueous porous media involves a process where the initial formation of diffusion layers subsequently becomes unstable with respect to fingering. We will present experimental examples of this transient growth process from mass diffusion to fingering instability using a pair of fluids that mimic the behavior of brine and carbon dioxide (brine is non-reactive in the experiments). In this experiment, performed in a Hele-Shaw cell, the permeability is adjusted by a back-lit diffuse illumination source. Pre-calibration of the transmitted light at various Potassium Permanganate concentrations allows us to convert the digital images into concentration fields and calculate the local and global flux in the system. We present preliminary results from this experimental study.
9:44AM AF.00009 CO₂ migration in saline aquifers: a gravity current model with capillary and solubility trapping¹, RUBEN JUANES, CHRISTOPHER MACMINN, MICHAEL SZULCZEWSKI, MIT — Injection of carbon dioxide into geological formations is widely regarded as a promising tool for reducing global atmospheric CO₂ emissions. While an accurate understanding of the post-injection spreading and migration of the plume of mobile CO₂ is essential, many aspects of the fundamental physics of CO₂ migration are poorly understood. Here, we develop a sharp-interface mathematical model for the post-injection migration of a CO₂ plume driven by groundwater flow in a sloping aquifer, subject to both residual trapping and CO₂ dissolution. We show that the interplay between dissolution and migration leads to three regimes, depending on how quickly the water beneath the plume saturates with dissolved CO₂. We develop some semi-analytical solutions to the migration equation when the water beneath the plume saturates very slowly or very quickly relative to plume motion, and we solve the migration equation numerically in general. We use these solutions to study the relative importance of capillary and solubility trapping, and the impacts of these physical mechanisms on the storage capacity of an aquifer.

¹Funded by the U.S. D.O.E. under Grant DE-FE0002041

Sunday, November 21, 2010 8:00AM - 10:10AM — Session AG GFD: Oceanography I Long Beach Convention Center 103B

8:00AM AG.00001 Internal wave emission by a stratified turbulent wake with non-zero net momentum, AMMAR ABDILGHANIE, PETER DIAMESSIS, Cornell University — The internal waves emitted by the stratified turbulent wake of a towed sphere are simulated using a 3D fully nonlinear spectral code in a parallel computing environment at two Reynolds numbers: 5,000 and 100,000 and three Froude numbers: 4, 16, and 64. The 2D Arc wavelet is used to extract the resonant horizontal scales from the horizontal divergence field on horizontal planes above the wake center line. Wave packets with lengths scales comparable to the sphere diameter are emitted from the wake with a decay rate increasing with both Froude and Reynolds numbers. The length scales increase with increasing Froude numbers and decreasing Reynolds numbers. Azimuth angles obtained from the Morlet2D wavelet are highly concentrated around 60 deg. Analysis of time series using 1D wavelet transforms reveals nearly constant frequencies (corresponding to polar angle 30±2 deg.) at the low Reynolds number simulations. At the high Reynolds number the polar angles are much higher (45-60 deg.) and slowly decay over time. Finally, wave steepness increases with both Reynolds and Froude numbers.

8:13AM AG.00002 Internal gravity wave absorption and reflection in a non-uniformly stratified Boussinesq fluid and subcritical Richardson number, MICHAEL RICHTER, AMMAR ABDILGHANIE, PETER DIAMESSIS, Cornell University — 2-D numerical simulations of the reflection and absorption of internal gravity waves in a non-uniformly stratified Boussinesq fluid are reported. The stratification profile combines a surface mixed layer separated from a uniformly stratified bottom layer (where waves are generated) by a sharp hyperbolic tangent pycnocline. The role of the incident wave steepness, the ratio of the vertical wavelength to the pycnocline thickness, and the peak pycnocline stratification strength in the reflection and absorption of the internal waves is studied. A hyperbolic tangent velocity profile collocated with the stratification profile is then introduced. The shear profile is such that a critical level exists inside the pycnocline and the gradient Richardson number is subcritical. Finally the influence of the stability of the shear flow on the wave reflection and absorption is appraised. Simulations are performed with and without an externally destabilized shear layer.

8:26AM AG.00003 The fluid mechanics of oil released into deep water, ANDREW W. WOODS, BP Institute, University of Cambridge — We present a series of models which describe the processes controlling the transport of oil through the water column from a deep-water release, accounting for the presence of dispersant which may cause the oil to break up into small droplets. We compare the model predictions with recent observations from the Gulf of Mexico in order to provide insights and constraints on the migration of oil through deep-water towards the surface.

8:39AM AG.00004 Non-invasive turbulent mixing across a density interface in a turbulent Taylor-Couette flow, C.P. CAULFIELD, BPI & DAMTP, U. of Cambridge, ANDREW W. WOODS, (BPI), J.R. LANDEL, (BPI & DAMTP), A. KÜESTERS, (BPI) — We present experimental measurements of the turbulent transport of salt across an interface between two layers of fluid of different salinities, confined to a cylindrical annulus with gap L where the inner cylinder rotates to produce an approximately irrotational mean azimuthal flow, with narrow boundary layers. We focus on the limit of high Richardson number flow, defined as \( Ri = \frac{\eta \Delta \rho H}{(\rho_0 u_{rms}^2)} \), where \( \rho_0 \) is a reference density, \( \Delta \rho \) is the time-dependent difference of the layers' mean densities, \( u_{rms} \) is the rms of the turbulent velocity fluctuations and \( H \) is the layer depth. The mean flow has \( Re \approx 10^3 \), and the turbulent fluctuations in the azimuthal and radial directions have rms speed of order 10% of the mean azimuthal flow. The interface between the two layers remains sharp, each layer remains well-mixed, and the vertical flux of salt between the layers, \( F_2 \approx (1.15 \pm 0.15) Ri^{-1} \Delta \rho H/L u_{rms} S \), where \( \Delta S \) is the spatially-averaged time-dependent salinity difference between the layers and \( A(H/L) \) is a function of the aspect ratio. The salt transport appears to be caused by turbulent eddies scarring and sharpening the interface and implies a constant rate of conversion of the turbulent KE to PE, independent of the density contrast between the layers.

8:52AM AG.00005 Ocean Circulation Modeling Using Adaptive Wavelet Collocation Method¹, SHANON RECKINGER, OLEG VASILYEV, University of Colorado at Boulder — The adaptive wavelet collocation method is applied to basin-scale, wind-driven ocean circulation models. This method solves the governing equations on temporally and spatially varying meshes, which allows higher effective resolution to be obtained with less computational cost. The grid adaptation is achieved by using the ability of wavelet multiresolution analysis to identify and isolate localized dynamically dominant flow structures, e.g., vortices, and to track these structures on adaptive computational meshes. In addition to studying how various ocean models behave on non-uniform, time varying grids, this work also sets out to improve the representation of continental topology and bottom bathymetry through an extension of the Brinkman penalization method. Due to the complicated geometry inherent in ocean boundaries, the stair-step representation used in the majority of current global ocean circulation models causes accuracy and stability problems. Brinkman penalization is a numerical technique used to enforce no boundary conditions through the introduction of a term to the governing equations. When coupled with the adaptive wavelet collocation method, the flow near the boundary can be well defined. This is especially useful for simulation of boundary currents. Therefore, the Gulf Stream and western boundary currents have been the focus of the work presented here.

¹This work was supported by DOE-CCPP (DE-FG02-07ER64168).
9:05AM AG.00006 Turning points for semidiurnal (M2) internal tides in the deep ocean. 
BENJAMIN KING, MARK STONE, HEPENG ZHANG, MICHAEL MARDER, HARRY SWINNEY, University of Texas at Austin, ROBERT SCOTT, National Oceanography Centre, Southampton — Previous work has mentioned the possibility of “turning points” in the deep ocean, depths at which the local buoyancy frequency $N(z)$ becomes smaller than the lunar semidiurnal (M2) tidal frequency: $N(z) < \omega_{M2}$ [W. Munk, Evolution of Physical Oceanography, MIT Press (1981)]. At these hypothetical locations, incident M2 internal tides would reflect from the turning points, resulting in regions in the deep ocean that are off limits to M2 internal tides. We have conducted the first systematic search for turning points by analyzing CTD (conductivity, temperature, depth) data obtained at 18,000 locations as a part of the World Ocean Circulation Experiment (WOCE), to determine $N(z)$ on a global scale. We have found that turning points are common in the deep ocean. We also present numerical simulations of internal wave beam interactions with turning points, and solutions of the vertical mode eigenvalue problem to determine what effects turning points might have on both internal wave beams and vertical modes.

9:18AM AG.00007 Langmuir circulation in the presence of lateral density gradients, KE LI, University of New Hampshire — Comparably little is known about the impact of lateral density gradients (associated with, e.g., submesoscale fronts) on Langmuir circulation in the ocean surface mixed layer. Here, 2D pseudospectral numerical simulations of the laterally stratified Craik-Leibovich (CL) equations are performed to elucidate the effect of an imposed horizontal density gradient on Langmuir cells. The dominant mode of instability consists of counter-rotating cells with up- and downwelling jets inclined to the vertical. Linear stability analysis confirms that although no instability occurs in the absence of the CL vortex torque, the dominant instability mode exhibits growth rates exceeding those realized in a constant density fluid. An energy budget is used to gain insight into the physics of this cooperative instability. The fully nonlinear simulations reveal a secondary instability, in which the tilted cells are laterally sheared, and a subsequent energy cascade to fine scales.

1GC gratefully acknowledges funding from NSF CAREER Award 0348981.
2Program in Integrated Applied Mathematics
3Mechanical Engineering Department

9:31AM AG.00008 Directional spreading of surface waves by wind-wave interaction, SANG SOO LEE, DAVID WUNDROW, Naval Surface Warfare Center, Carderock Division — A transition process from long-crested to short-crested wind-driven surface waves was analyzed using a first-principles-based asymptotic method. It is shown that a nonlinear interaction between wind and a surface wave, that initially grows linearly, can generate higher spanwise harmonics whose spanwise wave numbers are integer multiples of the primary wave. The amplitudes of the nonlinearly generated spanwise harmonics are of the same order as the primary fluctuation in the air and can be as large as the primary wave in the water. The mean wind is two-dimensional and there is no mean current. The primary wave can start as a single wave that propagates obliquely to the wind direction. The spanwise harmonics are generated by the nonlinear interaction in the air critical layer. They then induce corresponding perturbations in the water. Even though the magnitude of the primary surface wave is small, it generates spanwise harmonics of equal amplitude which lead to multi-directional water wave field.

1Supported by The Office of Naval Research and The National Science Foundation.

9:44AM AG.00009 Internal tide scattering (and generation) by arbitrary two-dimensional topography in arbitrary stratifications, THOMAS PEACOCK, MANIKANDAN MATHUR, MIT, GLENN CARTER, University of Hawaii — The generation and scattering of internal tides plays an important role in the energetics of the ocean. We have advanced the analytical Green function method to handle generation and scattering of internal tides by arbitrary two-dimensional topography in arbitrary stratifications. This provides a very useful tool for both fundamental studies of internal tide processes and for making reasonable predictions at important geophysical locations, such as the Hawaiian Ridge. Here, we give an overview of the method and present some fundamental and geophysical results obtained using it.

9:57AM AG.00010 ABSTRACT WITHDRAWN

Sunday, November 21, 2010 8:00AM - 10:10AM — Session AH Convection and Buoyancy Driven Flows I Long Beach Convention Center 103C

8:00AM AH.00001 Small-scale turbulent fluctuations beyond Taylor’s frozen flow hypothesis, PENGER TONG, XIAOZHOU HE, Department of Physics, Hong Kong University of Science and Technology, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences — The space-time cross-correlation function $C(r,t)$ of local temperature fluctuations in turbulent Rayleigh-Bénard convection is obtained from simultaneous two-point time series measurements. The obtained $C(r,t)$ is found to have the scaling form $C(r,t) = C(0,0) + [r(Ut)^2+(Vt)^2]$, where $U$ and $V$ are two characteristic velocities associated with the mean and rms velocities of the flow. The experiment verifies the theory and demonstrates its applications to a class of turbulent flows in which the requirement of Taylor’s frozen flow hypothesis is not met.

1Work supported by RGC of Hong Kong SAR and China NSFC.

8:13AM AH.00002 Role of Instability in State and Parameter Estimation of Rayleigh-Bénard Convection, ADAM PERKINS, MICHAEL SCHATZ, Georgia Institute of Technology — Predictive power in spatiotemporally complex systems is limited by several factors. Foremost among them is inherent system instability that can cause small initial uncertainty to grow rapidly. We address this issue in a Rayleigh-Bénard convection experiment, in which a novel technique of pattern control provides a tool for the repeatable imposition of a given convection pattern, e.g., a pattern near instability. Selected perturbations are applied to the reference pattern to create an ensemble of systems evolving from nearby initial conditions on both sides of the instability boundary. We deploy an efficient forecasting algorithm, the Local Ensemble Transform Kalman Filter (LETKF), to produce system state and parameter estimates from the convection patterns observed experimentally. Preliminary results of applying this state estimation algorithm to diverging pattern trajectories will be discussed.

1This work is supported by the National Science Foundation.
8:26AM AH.00003 Extensive Scaling from Computational Homology and Karhunen-Loève decomposition: Analysis of Rayleigh-Bénard Convection Experiments1, MICHAEI SCHATZ, HÜSEYIN KUR-TULDU, Georgia Institute of Technology, KONSTANTIN MISCHAikOW, Rutgers University — Spatiotemporally-chaotic dynamics in laboratory experiments on convection are characterized using a new dimension, $D_{\text{CH}}$, determined from computational homology. Over a large range of system sizes, $D_{\text{CH}}$ scales in the same manner as $D_{\text{KLD}}$, a dimension determined from experimental data using Karhuenen-Loeve decomposition. Moreover, finite-size effects (the presence of boundaries in the experiment) lead to deviations from scaling that are similar for both $D_{\text{CH}}$ and $D_{\text{KLD}}$. In the absence of symmetry, $D_{\text{CH}}$ can be determined more rapidly than $D_{\text{KLD}}$.

1Supported by DOE.

8:39AM AH.00004 Synchronization of Spatiotemporal Chaos in Rayleigh-Benard Convection1, ALIREZA KARIMI, MARK PAUL, Virginia Tech — We study the synchronization of spatiotemporal chaos in Rayleigh-Benard convection using numerical simulations of the Boussinesq equations. We consider one-way coupling between a principal and target convection domain. The principal domain is a large convection layer with no-slip boundaries on all material walls that is exhibiting spatiotemporal chaos. The target domain contains a convection layer that is smaller than the principal domain and is begun from random initial conditions in the temperature field. However, the sidewall boundary conditions of the target domain are given by the time dependent values of the principal domain at the equivalent location. The two domains are considered synchronized when the convection layers exhibit the same dynamics as measured by local and global diagnostics. Using this approach we quantify the length and time scales that describe the synchronization of the two domains over a variety of system parameters.

1NSF Award CBET-0747727, Virginia Tech’s Advanced Research Computing Center

8:52AM AH.00005 Does confined turbulent convection ever attain the ‘asymptotic scaling’ with 1/2-power?2, JOSEPH NIEMELA, International Centre for Theoretical Physics, KATEPALLI SREENIVASAN, New York University — We examine turbulent thermal convection for very high Rayleigh numbers using cryogenic helium in a cylindrical container with diameter-to-height aspect ratio $\gamma = 1$, and confirm that the Nusselt number, Nu, follows approximately the 1/3 power of the Rayleigh number, Ra, for $Ra \leq 2 \times 10^{14}$; $Nu = 0.064 Ra^{1/3}$. However, when Ra is pushed to higher values by approaching the critical point of helium in the temperature-pressure phase diagram, we observe a new state of enhanced heat transport, corresponding approximately to $Nu = 0.078 Ra^{1/3}$. The transition between the two states of the 1/3-power occurs with a log-log slope of roughly 1/2. Comparing experiments in the same apparatus but with $\gamma = 4$ - as well as slightly different paths through the pressure-temperature phase space with the same aspect ratio - we find that the transition value of Ra is not unique and can vary by an order of magnitude or more depending on experimental conditions. In particular, the transition does not correlate with dynamical parameters such as the Rayleigh number. However, it correlates reasonably well with a non-dimensional parameter related to variability of fluid conductivity and viscosity, occurring when the mean pressures and temperatures approach their critical values closer. No asymptotic transition to a half-power heat transport law was discerned.

9:05AM AH.00006 Transition to the ultimate regime in two-dimensional turbulent Rayleigh-Bénard convection1, RICHARD STEVENS, University of Twente, KAZUYASU SUGIYAMA, University of Tokyo, DETLEF LOHSE, University of Twente — The heat transfer in a RB system is determined by the Rayleigh number $Ra$ and the Prandtl number $Pr$. Various natural heat transfer phenomenon involve $Ra \gtrsim 10^{20}$ and thus extrapolations to this high Ra number regime are required. Here we present results from DNS for two-dimensional RBC with $Pr = 1$ in an aspect ratio $\Gamma = D/L = 0.23$, where $D$ and $L$ are the width and height of the box, respectively and achieve $Ra$ up to about $10^{14}$. For $Ra < 1 \times 10^{10}$ the Nusselt number varies nearly as the 1/3 power of $Ra$. However, for $Ra > 1 \times 10^{10}$ we find a sharp transition towards a regime where the Nusselt number varies nearly as the 1/2 power of $Ra$. A visualization of the simulation results reveals that the transition in the Nu number scaling are caused by a break-up of the large scale structures that are observed at lower Ra numbers.

1DEISA Extreme Computing Initiative (DECI-6), Stichting Fundamenteel Onderzoek der Materie (FOM).

9:18AM AH.00007 A low dimensional model for Rayleigh-Benard convection in rectangular domains1, JORGE BAILON-CUBA, JOERG SCHUMACHER, TU Ilmenau — A low dimensional model (LDM) for Rayleigh-Bénard (R-B) convection in rectangular boxes, based on the Galerkin projection of the Boussinesq equations onto a finite set of empirical eigenfunctions, is presented. The empirical eigenfunctions are obtained from Proper Orthogonal Decomposition (POD) of the field using the Snapshot Method. The most energetic POD modes give us a hint on the dynamic dominance of coherent flow patterns, and how well the original inhomogeneous flow can be modeled with a reduced number of modes. A quadratic non-homogeneous ODE system is obtained for the evolution of the modal amplitudes. A solution which considers the additional dissipation due to the neglected less energetic modes is considered in terms of a parameter $\epsilon \geq 0$, fixed at a value where the ensemble average of the total viscous and thermal dissipation in the model is the same as in the full simulation (DNS). We discuss first results of the evolution of the LDM and compare it with the DNS data of the R-B problem.

1This work is supported by the Deutsche Forschungsgemeinschaft (DFG).

9:31AM AH.00008 Measuring the departures from the Boussinesq approximation in Rayleigh-Bénard convection experiments1, HÜSEYIN KUR-TULDU, MICHAEL SCHATZ, Georgia Institute of Technology, KONSTANTIN MISCHAikOW, Rutgers University — Algebraic topology (homology) is used to characterize quantitatively non-Oberbeck-Boussinesq (NOB) effects in weakly turbulent Rayleigh-Bénard convection patterns from laboratory experiments. For fixed parameter values, homology analysis yields a set of Betti numbers that can be assigned to hot upflow and, separately, to cold downflow in a convection pattern. Analysis of data acquired under a range of experimental conditions where NOB effects are systematically varied indicates the difference between time-averaged Betti numbers for hot and for cold flow can be used as an order parameter to measure the strength of NOB-induced pattern asymmetries. This homology-based measure not only reveals NOB-effects that Fourier methods and measurements of pattern curvature fail to detect, but also permits distinguishing pattern changes caused by modified lateral boundary conditions from NOB pattern changes. These results suggest a new approach to characterizing data from either experiments or simulations where NOB effects are expected to play an important role.

1Supported by DOE.
importantly, certain compressible characteristics is still kept in the new model. The approach includes a dynamically scaling variable counting the overall thickness variation by viscosity. For compressible flow, we changed to use an inner product approach on compressible shear layers. To factor out the downstream viscous growth and then obtain models at lower dimension, our modified POD/Galerkin successfully used to obtain models at very low dimension for incompressible temporal shear layers (Wei and Rowley, 2009). In this study, we applied a similar

BASHAR QAWASMEH, MINGJUN WEI, New Mexico State University — A modified Proper Orthogonal Decomposition/Galerkin projection method has been open-loop model reduction applied to compressible flows. The proposed system reduction strategy is employed to shear layers respect momentum and energy balance equations in a mathematically rigorous manner — unlike unsteady Reynolds-averaged Navier-Stokes models or Smagorinsky-type reductions of the Navier-Stokes equation. The employed interaction models of lower order, based on a partition in slow, dominant and fast modes. In the reduced models, slow dynamics are incorporated as nonlinear manifold consistent with mean-field theory. Fast dynamics are stochastically treated and can be lumped in nonlinear eddy viscosity approaches. The employed interaction models allows substitution of the resulting modes into the PDEs, and Galerkin projection to ODEs. The approach is demonstrated for the flow over an open cavity at a moderate Reynolds number, where the actuator input occurs through blowing and suction through a small part of the boundary at the upstream upper part of the cavity.

1 Supported by AFOSR Grant FA9550-05-1-0411

8:13AM AJ.00002 System reduction strategy for Galerkin models of fluid flows1, MICHAEL SCHLEGEL, TU Berlin, Germany, BERND R. NOACK, Institut Pprime, Poitiers, France, MAREK MORZYNSKI, Poznan University of Technology, Poland, GILEAD TADMOR, Northeastern University, Boston, USA — We propose a system reduction strategy for control-oriented, spectral and Galerkin models of incompressible fluid flows. Key enabler is a finite-time thermodynamics (FTT) closure for the first and second moments. This decomposition allows substitution of the resulting modes into the PDEs, and Galerkin projection to ODEs. The approach is demonstrated for the flow over an open cavity at a moderate Reynolds number, where the actuator input occurs through blowing and suction through a small part of the boundary at the upstream upper part of the cavity.

1 Partially funded by the ANR Chaires d’Excellence program, the CNRS, the German Research Foundation (DFG), the US NSF and the US AFOSR.

8:26AM AJ.00003 Methods for the solution of very large flow-control problems that bypass open-loop model reduction1, PAOLO LUCHINI, Università di Salerno, THOMAS BEWLEY, University of California San Diego — The numerical discretization of the Navier-Stokes equations may easily lead to millions, or hundreds of millions, degrees of freedom. For the optimal control of such a problem, one is faced with either the solution of the full Riccati equation, numerically intractable for large systems, or with openloop model reduction, which may fail to capture the dynamics of interest. Here we present recent developments in our group about a third alternative: the Riccati-less solution of the unreduced optimal flow-control problem. These include a minimal-energy control algorithm based on the unstable eigenvectors alone, an iterative algorithm for the feedback kernel when the control is of much lower dimension than the state, and an iterative procedure for the leading eigenvalues and eigenvectors of the direct-adjoint Hamiltonian matrix that bypasses the solution of the Riccati equation.

1 Provincia di Salerno support is acknowledged for T. Bewley’s visiting scholarship in Salerno.

8:39AM AJ.00004 A least order model for temporally-developing compressible shear layers1, BASHAR QAWASMEH, MINGJUN WEI, New Mexico State University — A modified Proper Orthogonal Decomposition/Galerkin projection method has been successfully used to obtain models at very low dimension for incompressible temporal shear layers (Wei and Rowley, 2009). In this study, we applied a similar approach on compressible shear layers. To factor out the downstream viscous growth and then obtain models at lower dimension, our modified POD/Galerkin approach includes a dynamically scaling variable counting the overall thickness variation by viscosity. For compressible flow, we changed to use an inner product with both kinetic and thermal energy (Rowley, Colonius, Murray, 2004), then got the Galerkin model from the projection of the isentropic Navier-Stokes equations. The compressible model shows the capability to capture shear layer dynamics similarly but also slightly better than its incompressible version. More importantly, certain compressible characteristics is still kept in the new model.

1 Partially funded by the ANR Chaires d'Excellence program, the CNRS, the German Research Foundation (DFG), the US NSF and the US AFOSR.
natural frequencies. The corresponding Koopman modes display familiar physical features. Separated flow past a flat plate with an elliptical leading edge. Koopman analysis confirms the observation that such flows are dominated by a small set of the linearized (near-equilibrium), transient, and limit cycle (periodic shedding) dynamics. We also present a preliminary analysis of a high Reynolds number, decomposition (POD). Here we present a Koopman analysis of the flow past a 2-D cylinder. Using this single approach, we are able to identify modes relevant to the local dynamics. Each mode is associated with a distinct frequency, unlike those resulting from proper orthogonal decomposition (POD). The corresponding Koopman modes, which can be computed using an Arnoldi-like algorithm called Dynamic Mode Decomposition (DMD), provide a means of identifying structures relevant to the local dynamics. Each mode is associated with a distinct frequency, unlike those resulting from Proper Orthogonal Decomposition (POD). Here we present a Koopman analysis of the flow past a 2-D cylinder. Using this single approach, we are able to identify modes relevant to the linearized (near-equilibrium), transient, and limit cycle (periodic shedding) dynamics. We also present a preliminary analysis of a high Reynolds number, separated flow past a flat plate with an elliptical leading edge. Koopman analysis confirms the observation that such flows are dominated by a small set of natural frequencies.

The support of AFOSR through Grants FA9550-05-1-0369 monitored by Dr. Fariba Fahroo and FA9550-09-1-0189 managed by Dr. Douglas Smith is gratefully acknowledged.

9:05AM AJ.00006 A Comparison of Model Reduction Approaches for Feedback Control Design of Thermal Flows in Buildings. JEFF BORGGAARD, Virginia Tech, SUNIL AHUJA, United Technologies Research Center, JOHN BURNS, EUGENE CLIFF, Virginia Tech, AMIT SURANA, United Technologies Research Center — The application of distributed parameter control to spatiotemporal thermo-fluid systems requires the use of model reduction methods. The form of the optimal feedback control can inform design decisions, such as sensor and actuator selection and placement. A number of model reduction approaches for fluid systems have been put forward that are based on the proper orthogonal decomposition (POD). In this talk, we examine three approaches, the traditional POD-Galerkin model, the POD-Sensitivity model, and the Balanced-POD models. Our work is motivated by the building indoor environment control problem. Energy performance in building cooling and heating systems can be substantially improved by exploiting spatial temperature stratification and buoyancy that are prevalent in passive systems. We consider the control of airflow in a room with a passively cooled radiant ceiling and displacement ventilation provided near the room floor. For this problem, we approximate the full-order system with a reduced-order model-based control design.

9:18AM AJ.00007 Feedback control of the cylinder wake using balanced reduced order models. SIMON ILLINGWORTH, HIROSHI NAITO, KOJI FUKAGATA, Keio University — Feedback control is most successful when an accurate model of the system-to-be-controlled is available. For fluids, this can be achieved using a reduced order model which is balanced (meaning the input-output behaviour is properly captured). With this in mind, we consider feedback control of the cylinder wake in low Reynolds number simulations. Actuation is via blowing and suction on the cylinder’s surface, and a single velocity sensor in the wake is used. Balanced reduced order models are formed using the Eigensystem Realization Algorithm (ERA) at a number of Reynolds numbers. The reduced order models, validated by comparing their impulse responses to the full system, are then used in two ways. First, the “gain window” phenomenon seen in previous feedback control studies is reproduced (and therefore explained) by the models. We see that this gain window shrinks with increasing Reynolds number, the consequence being that feedback control with a simple proportional gain is not possible at higher Reynolds numbers. Second, loop shaping techniques are used to design “dynamic” controllers that are effective at higher Reynolds numbers, achieving complete suppression of vortex shedding at Reynolds numbers in excess of 100.

9:31AM AJ.00008 Feedback control of transition in boundary layer. ONOFRIO SEMERARO, SHERVIN BAGHERI, LUCA BRANDT, DAN S. HENNINGSON, Linne’ Flow Centre, KTH Mechanics - Stockholm (SWE) — We study the use of feedback control for the delay of laminar-turbulent transition in boundary layer. The mitigation of three-dimensional wavepackets of streaks and Tollmien-Schlichting waves is investigated numerically. The dynamics is studied from an input-output point of view: a set-up of spatially localized inputs (external disturbances and actuators) and outputs (sensor for the estimation and objective functions) is introduced for the control design. Sensors and actuators are distributed in arrays near the wall, spanning the homogeneous spanwise direction. Reduced-order models of the Navier-Stokes equations including the inputs and outputs, obtained via balanced truncation, are used to design an LQG controller. The controller provides an optimal signal that minimizes the amplitude of the perturbation downstream. Using a limited number of sensors and actuators (about 10-20 elements), the linear controller reduces substantially the energy growth of the instabilities arising in the boundary layer flows. In the final contribution, the mitigation of finite-amplitude perturbations in nonlinear simulations and the delay of the laminar-turbulent transition will be addressed.

9:44AM AJ.00009 Optimal actuator and sensor placement in the linearized complex Ginzburg-Landau system. KEVIN CHEN, CLARENCE ROWLEY, Princeton University — The linearized complex Ginzburg-Landau equation is a model for the evolution of small fluid perturbations, such as in a bluff body wake. We control this system by implementing actuators and sensors and designing an \( H_2 \)-optimal controller. We seek the optimal actuator and sensor placement that minimizes the \( H_2 \) norm of the controlled system, from flow disturbances to a cost on the perturbation and input magnitude. We formulate the gradient of the \( H_2 \) squared norm with respect to actuator and sensor positions, and iterate toward the optimal position. With a single actuator and sensor, it is optimal to place the actuator just upstream of the origin (e.g., the bluff body object) and the sensor just downstream. With multiple but an equal number of actuators and sensors, it is optimal to arrange them in pairs, placing actuators slightly upstream of sensors, and scattering pairs throughout the spatial domain. Global mode and Gramian analyses fail to predict the optimal placement; they produce \( H_2 \) norms about five times higher than at the true optimum. A wave maker formulation is better able to guess an initial condition for the iterator.

This work was supported by the DOD NDSEG Fellowship Program.

9:57AM AJ.00010 Analysis of Local Flow Dynamics Using Koopman Modes. JONATHAN TU, KEVIN CHEN, CLARENCE ROWLEY, Princeton University — The Koopman operator is a linear operator defined for any dynamical system, be it linear or nonlinear. The corresponding Koopman modes, which can be computed using an Arnoldi-like algorithm called Dynamic Mode Decomposition (DMD), provide a means of identifying structures relevant to the local dynamics. Each mode is associated with a distinct frequency, unlike those resulting from Proper Orthogonal Decomposition (POD). Here we present a Koopman analysis of the flow past a 2-D cylinder. Using this single approach, we are able to identify modes relevant to the linearized (near-equilibrium), transient, and limit cycle (periodic shedding) dynamics. We also present a preliminary analysis of a high Reynolds number, separated flow past a flat plate with an elliptical leading edge. Koopman analysis confirms the observation that such flows are dominated by a small set of natural frequencies. The corresponding Koopman modes display familiar physical features.

Supported by AFOSR grant FA9550-09-1-0257.
Sunday, November 21, 2010 8:00AM - 10:10AM –
Session AK Free Surface Flows: Spreading and Wetting Long Beach Convention Center 201B

8:00AM AK.00001 Surfactant-assisted spreading of an emulsion. ZHENZHEN LI, MATTHIEU ROCHE, Mechanical and Aerospace Engineering, Princeton University, ARNAUD SAINT-JALMES, Equipe Biophysique, Institut de Physique de Rennes, France, HOWARD A. STONE, Mechanical and Aerospace Engineering, Princeton University — We studied experimentally the spreading dynamics of a drop of a surfactant-stabilized oil-in-water emulsion over the free surface of a layer of solution of the same surfactant. The dynamics display three regimes. After the deposition of the emulsion drop, oil droplets are advected by a Marangoni flow, due to the difference in surfactant concentration between the emulsion and the liquid surface, and spread at the air/liquid interface. The oil droplets eventually stop, forming a dense ring, whose diameter is constant as long as oil droplets are transported by Marangoni flow. During this stage, oil droplets are moving rapidly on a surface with a low droplet concentration. Once the initial drop is empty, the ring collapses on itself, a phenomenon not yet reported experimentally to our knowledge. Spreading and retraction occur on a few hundred milliseconds while the ring stage lasts a few seconds. Using a laser sheet reflected by the surface of the liquid layer, we measured the shape of the surface and identified a jump of a few hundred microns in the layer thickness at the location of the ring. The existence of this jump points to hydrostatic pressure as the driving force for retraction. We also show that this system shares many features with other jumps.

8:13AM AK.00002 Quasi-Steady Capillary Driven Flow in Complex Geometries, MARK WEISLOGEL, ALEX BAKER, Portland State University, DONALD PETTIT, NASA Johnson Space Center — Lubrication theory has been successfully applied to a large class of local capillary driven flows along interior corners in simple conduits of uniform cross section (i.e. right polygonal cylinders). In this work, the evolution equations governing the local corner flows are re-scaled based on global conduit geometry, and the resulting system of equations are solved analytically in the quasi-steady limit. Several important closed form solutions are obtained with applications to passive filling, draining, and phase separations at both micro and macro length scales. Simple experiments are conducted that confirm the essential assumptions of the approach. As an example of the utility of the solutions, optimal geometries are computed for a unique “large length scale” flow aboard a spacecraft—a microgravity coffee cup.

8:26AM AK.00003 How cats lap. ROMAN STOCKER, PEDRO REIS, MIT, SUNGHWAN JUNG, Virginia Tech, JEFFREY ARISTOFF, Princeton University — We studied the lapping of the domestic cat (Felis catus) by combining high-speed photography with a laboratory model of lapping. We found that Felis catus laps by a subtle mechanism based on water adhesion to the dorsal side of the tongue and the creation of a liquid column, exploiting inertia to defeat gravity and pull liquid into the mouth. The competition between inertia and gravity controls the pinch-off time of the column, determining the optimal lapping frequency. J. Felis catus was found to operate near the optimum and theoretical analysis yielded a scaling, \( f \sim M^{-1/3} \), of lapping frequency with animal mass, \( M \). This prediction was verified by measuring lapping conduits across felid, from ocelots to lions, suggesting that the lapping mechanism is conserved among felines.

8:39AM AK.00004 Thermocapillary-assisted pulling of free liquid films, BENBOIT SCHEID, Universite Libre de Bruxelles, ERNST VAN NIEROP, Harvard University, HOWARD STONE, Princeton University — We study the formation of a free liquid film that is pulled out of a bath at constant speed and stabilized by the action of thermocapillary stresses prevailing at the free-surface. We show that only large shear induced by thermocapillary stresses allows for the stable pulling of the liquid film and that both extensional viscous stresses and gravity play no role. For small speeds and negligible inertia, the resulting thickness of the free film in independent on the pulling speed and proportional to the capillary length \( l_c \) as well as to a parameter \( f \) that measures the relative amplitude of the surface tension change at the interface. If this change is imposed (through a temperature gradient) along a distance \( d \) larger than the characteristic length \( l = l_c \sqrt{2f} \) of the system, the film thickness decreases with increasing \( d \); otherwise it is independent of \( d \). For large speeds and non-negligible inertia, the film thickness decreases with an increase of the Weber number. We also show how the results depend on heat transfer properties. The present theory suggests that very thin ribbons or foils of molten material can be drawn out of a melt over a wide range of thicknesses at speeds relevant to manufacturing.

8:52AM AK.00005 Increasing the critical speed of wetting failure through meniscus confinement. ERIC VANDRE, SATISH KUMAR, University of Minnesota, MARCIO CARVALHO, Pontificia Universidade Catolica do Rio de Janeiro, Brazil — Dynamic wetting is a crucial step of fluid-fluid displacement along a solid surface, such as the deposition of a coating liquid onto a moving substrate. At some critical process speed wetting fails and the displaced phase (e.g. air) is entrained within the displacing phase. Improving upon current industrial production methods, though its influence remains unclear. In this study, we explore the effects of confinement on wetting failure with a laboratory-scale plunge-coating system. Our experimental apparatus consists of a steel roll that plunges into a bath of glycerol. Confinement is imposed by bringing a coating die near the wetting line, and liquid is injected through the die to compensate for liquid being dragged away with the roll. Flow visualization is used to record the critical roll speed at which wetting failure occurs. The data show a clear increase in the critical speed with increasing confinement. A model based on the lubrication approximation does a remarkable job in accurately predicting the increase in the critical speed relative to the unconfined value.

9:05AM AK.00006 Inertial and three-dimensional effects in stretching liquid bridges near plates and cavities. SHAWN DODDS, University of Minnesota, MARCIO CARVALHO, Pontificia Universidade Catolica do Rio de Janeiro, SATISH KUMAR, University of Minnesota — The dynamics of liquid bridges are relevant to a wide variety of applications, including high-speed printing and extensional rheometry. Analysis of these systems is often performed assuming axisymmetric Stokes flow, although in printing processes these assumptions may not always hold. To address this issue, we use the finite element method to study the stretching of a finite volume of liquid between two surfaces in two model problems. In the first problem, we consider an axisymmetric liquid bridge between a stationary cavity and a moving flat plate. The contact lines are allowed to slip, and centrifugal effects can develop in the curved part of the stream. We give a quantitative criterion for the onset of meandering, and confirm it by comparing to the flow of a rivulet between to glass plates which are wetted completely. Above the threshold, the rivulet follows a irregular pattern with a typical wavelength of a few centimeters. This very general mechanism should hold in very different situations (inclined or vertical plates, total or partial wetting, pure fluid or surfactants...) provided that pinning effects of contact lines are not too strong.

9:18AM AK.00007 A general mechanism for the meandering of rivulets. NADINE VALADE, ADRIAN DAERR, Laboratoire Matiere et Systemes Complexes (MSC), UMR 7057 of CNRS and Paris Diderot University, Paris, France, JENS EGGERS, School of Mathematics, University of Bristol, Bristol BS8 1TW, United Kingdom, LAURENT LIMAT, Laboratoire Matiere et Systemes Complexes (MSC), UMR 7057 of CNRS and Paris Diderot University, Paris, France — A rivulet flowing down an inclined or vertical plane often does not follow a straight path, but starts to meander. We found that this instability can appear from two key ingredients: fluid inertia and anisotropy of the friction between rivulet and substrate. Meandering occurs if the fluid inertia normal to the instantaneous flow direction is more difficult than parallel to it. This slows down the downstream motion of a meander with respect to that of the fluid, and centrifugal effects can develop in the curved part of the stream. We give a quantitative criterion for the onset of meandering, and confirm it by comparing to the flow of a rivulet between to glass plates which are wetted completely. Above the threshold, the rivulet follows a irregular pattern with a typical wavelength of a few centimeters. This very general mechanism should hold in very different situations (inclined or vertical plates, total or partial wetting, pure fluid or surfactants...) provided that pinning effects of contact lines are not too strong.
Flow transition behavior between the film flow and rivulet flow on an inclined wall 

YOSHIYUKI ISO, IHI Inc., XI CHEN, Columbia University — Gas-liquid two-phase flows on the wall like liquid film flows, which are the so-called wetted wall flows, are observed in many industrial processes such as absorption, desorption, distillation and others. For the optimum design of packed columns widely used in those kind of processes, the accurate predictions of the wetted wall flow behavior in packing elements are important, especially in order to enhance the mass transfer between the gas and liquid and to prevent flooding and channeling of the liquid flow. The present study focused on the effects of the change of liquid flow rate and the wall surface texture treatments on the characteristics of wetted wall flows which have the drastic flow transition between the film flow and rivulet flow. In this paper, gas-liquid two-phase flow simulation by using the volume of fluid (VOF) model is applied into wetted wall flows. Firstly, present results showed that the hysteresis of the flow transition between the film flow and rivulet flow arose against the increasing or decreasing stages of the liquid flow rate. It was supposed that this transition phenomenon depends on the history of flow pattern as the change of curvature of interphase surface which leads to the surface tension. Secondary, the present simulations showed that surface texture treatments added on the wall can improve the prevention of liquid channeling and can increase the wetted area.

Supported by the Marie Curie MULTIFLOW Network, by ESA & BELSPO PRODEX projects, and by FRS-FNRS.

Experimental measurements of contact angles with evaporation by interferometry 

JULIEN SEBILLEAU, SAM DEHAECK, PIERRE COLINET, TIPS, Université Libre de Bruxelles — Volatile liquids, on a substrate under total wetting conditions, exhibit an apparent contact angle even in the case of a static contact line. This contact angle is linked to the evaporation process that induces a (micro)flow in the contact line region. We study experimentally this contact angle for liquids evaporating into ambient air, in the case of a meniscus generated at the top of a Hele-Shaw cell, the two glass walls of which being placed at different heights. The shape of this meniscus is then recorded with two kinds of interferometers (Mach-Zehnder in transmission, and reflection interferometry), which allow an accurate measurement of the apparent contact angle at some distance from the actual contact line. Both static and moving (advancing or receding) contact lines situations are studied and several liquids are used. For advancing contact lines, instabilities leading to droplets formation are also observed.

Supported by the Marie Curie MULTIFLOW Network, by ESA & BELSPO PRODEX projects, and by FRS-FNRS.

Aeroacoustic sources in phonation 

MICHAEL KRANE, ARL Penn State — An analysis of the flow through human vocal folds is used to identify the primary aerosound source mechanisms in human vocalization. The acoustic fields on either side of the constriction are matched using equations describing the flow through the constriction. The form of the resulting sound fields indicates that the primary source of sound is the unsteady aerodynamic drag due to separated flow, and that secondary sources arise from changes in glottis volume and the movement of the separation point. The source strengths are shown to depend on the incident sound field, calling into question the “source-filter” theory of voice production. A control volume analysis supports these results.

Acknowledgment support of NIH.

The impact of asymmetrical flows on pathological speech 

BYRON D. ERATH, George Washington University, SEAN D. PETERSON, University of Waterloo, MICHAEL W. PLESNIAK, George Washington University — In voiced speech the vocal folds form a divergent glottal passage during the closing phases of the phonatory cycle. Due to the adverse pressure gradient, asymmetric flow develops within the glottis causing the glottal jet to separate from one vocal fold wall, and fully-attach to the opposing wall. The asymmetric pressures that arise from this flow configuration directly influence the vocal fold energy exchange process, and are expected to have the greatest influence on vocal fold motion when pathologies that affect the vocal fold musculature are present. A theoretical flow solution that produces the pressure distributions arising from asymmetric glottal flows is implemented into a two-mass model of speech. The impact of flow asymmetries on pathological vocal fold motion is investigated by modifying the tissue parameters of the speech model to represent unilateral paralysis. The influence of asymmetric flow behavior on pathological vocal fold motion is quantified and compared to the commonly-reported simplified case involving symmetric flow behavior.

Supported by the National Science Foundation under Grant No. CBET-1036280.

Effect of subglottic stenosis on the flow-induced vibration of a self-oscillating computational vocal fold model 

SIMEON L. SMITH, SCOTT L. THOMSON, Brigham Young University Mechanical Engineering — The subglottis plays an important role in voice production; however, in general the role of subglottal geometry in phonation is not well understood. This research focuses on studying how subglottic stenosis, or a narrowing of the airway below the vocal folds, affects the response of a self-oscillating computational vocal fold model. Methods are described for computational model development, including stenotic geometry definition from CT scan images, incorporation of the stenosis into a finite element fluid-structure interaction model, and parametric variation of the degree of stenosis severity. Results are presented for a normal (no stenosis) case and five cases of varying degrees of stenosis severity. Qualitative and quantitative comparisons of vocal fold vibratory motion and of flow behavior for the six cases are made, including characterization of flow patterns in the subglottis, glottal width and flow rate time histories, vibration frequency, and airflow resistance.

Examination of Flow in a Scaled-Up Vocal Fold Model for Diseased Conditions 

ERICA SHERMAN, LUCY ZHANG, WANG XINSHI, WEI TIMOTHY, RPI, MICHAEL KRANE, Penn State ARL — An experiment to provide DPIV measurements in a scaled up dynamic human vocal fold model is presented. The 10x scale vocal fold model is a new design that incorporates both the rocking as well as the oscillatory open/close motions of vocal fold motions. The experiment is run in a free-stream water tunnel where the oscillation frequencies and flow speeds are dynamically matched to physiologic conditions for both male and female phonation. The effects associated with vocal fold paralysis will be discussed. Flow measurements showing fluid kinematics including jet velocity and orientation, and vortex shedding as a function of time through an oscillation cycle will be presented. In addition, key data relevant to phonation, such as volumetric flow rate and glottal behavior will be presented.
8:52AM AL.00005 Numerical study of the fluid-solid interactions in human vocal folds using finite element method  
XINGSHI WANG, ERICA SHERMAN, MICHAEL KRANE, TIMOTHY WEL, LUCY ZHANG  
The goal of this study is to investigate the motion and deformation of human vocal folds during phonation using finite element method. The voice process is a fluid-structure interaction problem and it is also a self-oscillated system induced by the airflow with constant pressure difference. Here, the vocal folds are modeled with 2-D hyperelastic structures embedded in a channel with applied constant pressure difference at the inlet and outlet to represent the lung pressure. Our fully coupled fluid-structure interaction numerical method can capture the open/close process and the deformed shape of the vocal folds based on the given pressure. From the numerical results, we are able to capture the periodic features for the variables of interest at the throat, such as volume flow rate, velocity and pressure. These dynamic variable outputs may assist us to perform further energy balance analysis to fully understand the physical mechanisms of normal and disordered phonation.

9:05AM AL.00006 Experimental study of the aeroacoustic-aeroelastic behavior of model vocal folds  
ELIZABETH CAMPO, ARL Penn State, ERNESTO CAMARENA, Purdue University, MICHAEL KRANE, ARL Penn State  
The effect of vocal fold body stiffness and bilateral asymmetry was studied using a life-size physical model of the human airway using interchangeable silicone rubber models of the human vocal folds. The two layer vocal fold models are comprised of an inner body layer and an outside cover layer. The following measures were used to assess the effect of body stiffness and asymmetry: radiated sound power, phonation threshold pressure and aeroacoustic source strengths. Results obtained from the human airway model compared favorably with behavior observed in human subjects. Furthermore, the results reveal that the asymmetric cases required a higher subglottal pressure to initiate phonation and radiated less intense sound, in comparison to the symmetrical configuration.

1Acknowledgment support from NIH and PSU-ARL Water Tunnel summer intern program.

9:18AM AL.00007 Bird song: in vivo, in vitro, in silico  
ARYESH MUKHERJEE, SHREYAS MANDRE, LAKSHMINARAYAN MAHadevAN, Harvard University, BIRD SONG TEAM  
Bird song, long since an inspiration for artists, writers and poets also poses challenges for scientists interested in dissecting the mechanisms underlying the neural, motor, learning and behavioral systems behind the beak and brain, as a way to recreate and synthesize it. We use a combination of quantitative visualization experiments with physical models and computational theories to understand the simplest aspects of these complex musical boxes, focusing on using the controllable elastohydrodynamic interactions to mimic aural gestures and simple songs.

9:31AM AL.00008 Thin Film Evolution Over a Thin Porous Layer: Modeling a Tear Film on a Contact Lens  
DANIEL ANDERSON, KUMNIT NONG, George Mason University  
We examine a mathematical model that describes the behavior of the pre-contact lens tear film of a human eye. Our work examines the effect of contact lens thickness and lens permeability and slip on the film dynamics. A mathematical model for the evolution of the tear film is derived using a lubrication approximation applied to the hydrodynamic equations of motion in the fluid film and the porous layer. The model is a nonlinear fourth order partial differential equation subject to boundary conditions and an initial condition for post-blink film evolution. We find that increasing the lens thickness, permeability and slip all contribute to an increase in the film thinning rate although for parameter values typical for contact lens wear these modifications are minor. The presence of the contact lens can, however, fundamentally change the nature of the rupture dynamics as the inclusion of the porous lens leads to rupture in finite time rather than infinite time.

1Sponsored by the NSF (DMS-0639300, DMS-0709095).

9:44AM AL.00009 A Model for the Precorneal Tear Film with Osmolarity and Corneal Supply  
R.J. BRAUN, University of Delaware, P.E. KING-SMITH, The Ohio State University  
In the human tear film, a thin liquid layer is spread with a blink; it subsequently levels due to surface tension and evaporates more slowly than pure water due to the floating lipid layer. While eventually the tear film almost always ruptures, recent evidence suggests that in some cases supply of fluid from the cornea or conjunctiva may prolong the life of the tear film and prevent the osmolarity (combined concentration of certain salts and sugars) from reaching very large values that can cause irritation and damage. We incorporate osmolarity into a lubrication model for the tear film and study the dynamics of the tear film with osmotic supply from the corneal surface by numerically solving equations for the film thickness and osmolarity. It is treated as a classical osmotic semi-permeable barrier with parameters appropriate to the cornea. The tear film thinning may be slowed by these effects, and in some cases rupture prevented. The value of the osmolarity in regions thinned by evaporation is reduced by osmosis from the underlying surface.

9:57AM AL.00010 Viscoelastic Properties of Vitreous Gel  
H. PIrouz KAVEHPOUR, POORIA SHARIF-KASHANI, University of California, Los Angeles  
We studied the rheological properties of porcine vitreous humor using a stressed-control shear rheometer. All experiments were performed in a closed environment at body temperature to mimic in-vivo conditions. We modeled the creep deformation using a two-element retardation spectrum model. By associating each element of the model to an individual biopolymeric system in the vitreous gel, a separate response to the applied stress was obtained from each component. The short time scale was associated with the collagen structure, while the longer time scale was related to the microfibrils and hyaluronan network. We were able to distinguish the role of each main component from the overall rheological properties. Knowledge of this correlation enables us to relate the physical properties of vitreous to its pathology, as well as optimize surgical procedures such as vitrectomy.

Sunday, November 21, 2010 8:00AM - 10:10AM  
Session AM Microfluids: General I: Electrokinetic  
Long Beach Convention Center 202B

8:00AM AM.00001 Colloidal particle motion in micro galvanic reactors through tailored electrokinetic fluid flow  
LINDA JAN, CHRISTIAN PUNCKT, Princeton University, Princeton, NJ 08544, BORIS KHUSID, New Jersey Institute of Technology, Newark, NJ 07102, ILHAN A. AKSAY, Princeton University, Princeton, NJ 08544  
Using an array of galvanic micro electrodes (e.g., anodic copper and cathodic gold) in contact with an acidic colloidal suspension, we have previously demonstrated autonomous control of particle trajectory and the location of particle deposition which affected the crystallinity of 2D colloidal crystals on the anodes. Particle velocities and the locations of initial particle deposition are affected by the electrode geometry and reaction time. We now present data on the effects of geometry and time on the copper dissolution rate and the associated electrokinetic phenomena. Particle velocities increase with the copper dissolution rate and the steepness of its lateral variation. Experiments and theoretical results reveal that the different location of deposition is related to the difference in the lateral gradient of the dissolution rate.
8:13AM AM.00002 Suppression of Electrokinetic Flows by Surface Roughness1, ROBERT MESSINGER, TODD SQUIRES, University of California, Santa Barbara — In microfluidic systems, electro-osmotic flows are a promising alternative to mechanical pressure-driven flows, since electrokinetic flow rates are independent of microchannel dimensions and may enable the design of portable (e.g., battery-operated) devices. We show that nanoscale surface roughness, which commonly occurs on microfabricated metal electrodes, can significantly suppress electro-osmotic flows when excess surface conductivity is appreciable. We demonstrate the physical mechanism of electro-osmotic flow suppression due to surface curvature, compute the effects of varying surface conductivity and roughness amplitude on the slip velocities of a model system, and identify scalings for flow suppression in different regimes of surface conduction. We suggest that surface roughness may be one factor that accounts for large discrepancies between classical electrokinetic theory and modern microfluidic experiments.

1This work was supported by the UCSB/LANL Institute of Multi-scale Materials Studies

8:26AM AM.00003 Note on the Nonlinear Electrokinetic Effects in Microchannel Flow: Exact Analytical Solutions for Sinh-Poisson Equation, ALAN CHENG HOU TSANG, KWOK WING CHOW, Department of Mechanical Engineering, The University of Hong Kong, Hong Kong, China — Electrokinetic effects are important phenomena for fluid flow in microchannels, especially in mechanical systems involving movable micromechanical devices. Electrokinetic effects arise from electric double layer, which is a layer of charges attached to the dielectric surfaces as a result of the interaction of charges between ionized solution and dielectric surfaces. Electric potential inside the flow field is governed by the nonlinear Poisson-Boltzmann equation. Owing to the difficulty in solving the nonlinear equation, Debye - Hückel approximation, having an assumption of small electric potential, is a common approach to solve the linearized problem. In the present work, exact analytical expressions are obtained for the fully nonlinear sinh - Poisson equation without invoking the linear approximation. These solutions give insight on treating flow problems when Debye - Hückel approximation does not hold. Selected examples of solutions for a rectangular cell with zero homogenous boundary conditions applied on three wall surfaces are used for comparisons between the fully nonlinear and the linearized cases. Significant discrepancies are observed if the potential is not small, hence the present nonlinear theory is essential to better describe the physics involved.

8:39AM AM.00004 The electrokinetics of near-wall colloidal particles measured by evanescent-wave particle velocimetry1, YUTAKA KAZOE, University of Tokyo, MINAMI YODA, Georgia Institute of Technology — Understanding the near-wall dynamics of suspended colloidal particles subject to electric fields is of interest in microfluidics. Most previous colloid science studies using total internal reflection microscopy to study these dynamics have considered a single particle in a quiescent fluid. We instead analyze the dynamics of an ensemble of fluorescent particles illuminated by evanescent waves using multilayer nanoparticle tracking velocimetry (MnPTV). The technique exploits the decay of the evanescent-wave intensity with wall-normal distance z to extract near-wall particle z-distributions and flow velocities at different distances from the wall. Here electrokinetically driven flows through ~40 μm deep fused-silica channels are studied using MnPTV. The results for the particle distributions near the wall are highly nonuniform due to electrostatic and van der Waals effects, and that the distributions vary with both the applied electric field E and a due to forces that scale as E2 and a2. Despite this variation in the near-wall particle distributions, the MnPTV results give Brownian diffusion coefficients that agree with theoretical predictions and the uniform velocity profiles typical of electroosmotic flow.

1Supported by JSPS and NSF.

8:52AM AM.00005 Secondary flow effect on electrokinetic transport in curved channels and microfluidic mixing, MYUNG-SUK CHUN, JIN-MYEONG LIM, Korea Institute of Science and Technology (KIST) — This presentation reports the numerical framework and important new results regarding the velocity pattern, vorticity, and mixing property, with variations of channel geometry and heterogeneity of surface properties. Extending our previous studies, secondary Dean flow in curved rectangular microchannels is examined by applying the finite volume/SIMPLE algorithm for the pressure-driven electrokinetic transport coupled with the Poisson-Boltzmann/Navier-Stokes/Nernst-Planck equations. Hydrophilic glass and hydrophobic polymers with fluid slip are combined to create different channel configurations with ranging complementary aspect ratios. Simulation results show that, contrary to the case of narrow-bore channels, the streamwise axial velocity tends to shift toward the inner wall caused by a stronger effect of the spanwise pressure gradient. We observe the presence of pairs of counter-rotating vortices perpendicular to the flow direction and evaluate the circulation magnitude. The increasing rate of inner shift with increasing curvature ratio is more significant in the shallow channel, and the patterns of axial velocity and vorticity alter by the heterogeneity effect of surfaces occupying a large area. In addition, the inertial force should be considered for precise control of the micro- or nanoflows.

9:05AM AM.00006 Electrokinetic locomotion due to Reaction Induced Charge Auto-Electrophoresis, JEFFREY MORAN, JONATHAN POSNER — Synthetic nanomotors, like their biological counterparts, propel themselves through aqueous solutions by harvesting chemical energy from their local environment and converting it to mechanical energy. We study bimetallic rod-shaped particles which move autonomously by catalytically decomposing hydrogen peroxide to oxygen and water. We present a scaling analysis and computational simulations that describe the locomotion of bimetallic rod-shaped motors in hydrogen peroxide solutions due to reaction-induced charge auto-electrophoresis. The model shows that the locomotion results from electrical body forces in the surrounding fluid, which are generated by a coupling of an asymmetric dipolar charge density distribution and the electric field it generates. The simulations make the predictions, in agreement with experiment, that the rods’ velocity depends linearly on both the surface charge and reaction rate.

9:18AM AM.00007 Electrophoretic mobility of deformable elastic particles in confined geometries, TONG GAO, HOWARD HU, University of Pennsylvania — Electrophoretic motion of a dielectric neo-Hookean elastic particle in a confined microchannel is simulated by an Arbitrary Lagrangian-Eulerian moving mesh technique. The particle with a fixed zeta potential is initially elliptical and aligned perpendicular to the direction of the applied electric field. The size of the electrical double layer is assumed to be negligible compared with the particle size and the classical Helmholtz-Smoluchowski slip boundary conditions are applied on the particle surface. When the Reynolds number is low, the elastic deformation is purely induced by the viscous shear force distribution along the body. In the unbounded domain, it is known that the particle will move with a constant Helmholtz-Smoluchowski velocity which is independent of the particle deformation. However, in the confined channel, the rigid walls not only alter the particle-electrical field interaction but also tend to slow the particle motion. To explore the wall effect on the electrophoretic mobility of the particle, the migration velocity is examined by systematically changing both the channel size and the material properties. Also the particle motion in non-Newtonian fluids are simulated and compared with Newtonian cases.
9:31AM AM.00008 Lateral Migration and Three-dimensional Focusing of Particles in Microchannel Electrophoresis1, LITAO LIANG, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921, SHIZHI QIAN, Department of Aerospace Engineering, Old Dominion University, Norfolk, VA 23529, USA, XIANGCHUN XUAN, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921 — The fundamental study of particle electrophoresis in microchannels is relevant to many applications. It has been accepted that particles move parallel to the applied electric field in a straight uniform microchannel. In this talk we present the first experimental demonstration of lateral particle migration in electrophoresis through a rectangular microchannel. This phenomenon is due to the electrical force induced by the asymmetric electric field around the particle near a channel wall. We demonstrate that such cross-stream particle motion in electrophoresis can focus neutrally buoyant particles to the centerline of a rectangular microchannel. This three-dimensional electrophoretic particle focusing may potentially be used in micro flow cytometers.

9:44AM AM.00009 Highly non-linear induced-charged electroosmosis for flat electrodes, GAURAV SONI, CARL MEINHART, Dept. Mech. Eng., University of California, Santa Barbara, CA 93106, MATHIAS B. ANDERSEN, HENRIK BRUUS, Dept. Micro- and Nanotechnology, Technical University of Denmark, 2800 Kgs. Lyngby, Denmark — We have simulated induced-charged electroosmotic flow over a flat surface in the highly nonlinear regime, where the notion of the double layer as a linear capacitor is invalid. We have developed two completely independent solution methods: one resolving the double layer under continuum assumption and the other treating the double layer as an effective boundary condition balancing the normal and tangential flux of ions. We show that tangential transport within the double layer leads to gradients in bulk scalar fields. By comparing the effective-boundary model with continuum model, we are able to quantify the accuracy of the effective boundary model. There are certain simplifications of the effective boundary model. One simplification ignores the bulk concentration gradients but incorporates surface conduction. Another simplification is based on linearization and ignores both bulk concentration gradients and surface conduction. We quantify the accuracy of these simplifications.

9:57AM AM.00010 Nonlinear waves in electromigration dispersion1, SANDIP GHOSAL, ZHEN CHEN, Northwestern University — Electromigration dispersion occurs in CE when sample concentrations are sufficiently high. The signal is known to exhibit features such as sharp concentration “shocks” that are reminiscent of nonlinear waves. We consider a simplified 3 ion model consisting only of strong electrolytes that are equi-diffusive. The sample concentration is then shown to obey a one dimensional advection diffusion equation with a concentration dependent advection velocity which reduces to Burgers’ equation if the sample loading is not too high. Thus, the time dependent problem is exactly solvable with arbitrary initial conditions and in the case of small diffusivity concentration shocks are formed. Analytical formulas are derived for the shape, width, and migration velocity of the sample peak and it is shown that axial dispersion at long times may be characterized by an effective diffusivity that is exactly calculated.

1Supported by the NIH under grant R01EB007596.

Sunday, November 21, 2010 8:00AM - 9:57AM — Session AN Vortex Dynamics and Vortex Flows I Long Beach Convention Center 202C

8:00AM AM.00001 A comparison of vortex and pseudo-spectral methods at high Reynolds numbers, ANTHONY LEONARD, GALCIT, California Institute of Technology, WIM VAN REES, PETROS KOUMOUTSAKOS, Chair of Computational Science, ETH Zurich, CH-8092, Switzerland — We validate the hybrid particle-mesh vortex method against a pseudo-spectral method in simulations of the Taylor-Green vortex and colliding vortex tubes at Re = 1600 - 10,000. The spectral method uses the smooth filter introduced in [1]. In the case of the Taylor-Green vortex, we observe very good agreement in the evolution of the vortical structures albeit small discrepancies in the energy spectrum only for the smallest length scales. In the collision of two anti-parallel vortex tubes at Re = 10,000, there is very good agreement between the two methods in terms of the simulated vortical structures throughout the first reconnection of the tubes. The maximum error in the effective viscosity is below 2.5% and 1% for the vortex method and the pseudo-spectral method respectively. At later times the agreement between the two methods in the vortical structures deteriorates even though there is good agreement in the energy spectrum. Both methods resolve an unexpected vortex breakdown during the second reconnection of the vortex tubes.


8:13AM AN.00002 Vortex methods with immersed lifting lines applied to LES of wind turbine wakes, PHILIPPE CHATELAIN, LAURENT BRICTEUX, GREGOIRE WINCKELMANS, Universite catholique de Louvain (UCL) - Institute of Mechanics, Materials and Civil Engineering (IMMC), PETROS KOUMOUTSAKOS, Chair of Computational Science - ETH Zurich — We present the coupling of a vortex particle-mesh method with immersed lifting lines. The method relies on the Lagrangian discretization of the Navier-Stokes equations in vorticity-velocity formulation. Advection is handled by the particles while the mesh allows the evaluation of the differential operators and the use of fast Poisson solvers. We use a Fourier-based fast Poisson solver which simultaneously allows unbounded directions and inlet/outlet boundaries. A lifting line approach models the vorticity sources in the flow. Its immersed treatment efficiently captures the development of vorticity from thin sheets into a three-dimensional field. We apply this approach to the simulation of a wind turbine wake at very high Reynolds number. The combined use of particles and multiscale subgrid models allows the capture of wake dynamics with minimal spurious diffusion and dispersion.

8:26AM AN.00003 Vortex methods for fluid-structure interaction problems with deforming geometries and their application to swimming, MATTIA GAZZOLA, Chair of Computational Science - ETH Zurich, PHILIPPE CHATELAIN, Univ catholique de Louvain (UCL) - Institute of Mechanics, Materials and Civil Engineering (IMMC), PETROS KOUMOUTSAKOS, Chair of Computational Science - ETH Zurich — We present a vortex particle-mesh method for fluid-structure interaction problems. The proposed methodology combines implicit interface capturing, Brinkmann penalization techniques, and the self-consistent computation of momentum transfer between the fluid and the structure. In addition, our scheme is able to handle immersed bodies characterized by non-solenoidal deformations, allowing the study of arbitrary deforming geometries. This attractively simple algorithm is shown to accurately reproduce reference simulations for rigid and deforming structures. Its suitability for biological locomotion problems is then demonstrated with the simulation of self-propelled anguilliform swimmers.

8:39AM AN.00004 A Lagrangian Vortex Method for the Barotropic Vorticity Equation on a Rotating Sphere1, ROBERT KRASNY, LEI WANG, University of Michigan — We present a Lagrangian vortex method for the barotropic vorticity equation on a rotating sphere. The method solves for the flow map using Lagrangian particles and panels. The velocity is computed by evaluating the Biot-Savart integral on the sphere. An adaptive refinement strategy is implemented to maintain accuracy over long times. Results are presented for propagating Rossby-Haurwitz waves and localized vortex interactions.

1Supported by NSF grant ATM-0723440.
The vortex ring results show similar flow structures observed qualitatively in the lower Re rings were generated with a piston-cylinder vortex ring generator using piston stroke-to-diameter ratios and Re vortex ring which was not observed for lower \( \phi \) interacting with an approximately fixed number of screens for low \( \phi \) smaller vortical structures after its interaction with the first screen. For the \( \phi \), MUSTAFA N. MUSTA, Southern Methodist University, PAUL S. KRUEGER, Southern Methodist University — DPIV measurements of the present research will provide a tool for designing new bridges and retrofitting old ones.

The CFD results matched the experimental data in terms of the relationship between inundation ratio and force measured at the bridge. The results of the study showed that the streamlined deck significantly reduces drag, lift, and moment coefficient in comparison to the other bridge deck types. The CFD results matched the experimental data in terms of the relationship between inundation ratio and force measured at the bridge. The results of the present research will provide a tool for designing new bridges and retrofitting old ones.

Highway Research Center establish the foundations of validated computational practices to address the research needs of the transportation community. Three bridge deck prototypes were used: a typical six-girder highway bridge deck, a three-girder deck, and a streamlined deck designed to better withstand the hydraulic forces. Results of the study showed that the streamlined deck significantly reduces drag, lift, and moment coefficient in comparison to the other bridge deck types. The CFD results matched the experimental data in terms of the relationship between inundation ratio and force measured at the bridge. The results of the present research will provide a tool for designing new bridges and retrofitting old ones.

The authors would like to acknowledge the support of AFOSR under grant FA9550-08-1-0406

Sunday, November 21, 2010 8:00AM - 10:10AM – Session AP Nanofluids I Long Beach Convention Center 203A

8:00AM AP.00001 Experimental Study of Fluid Structure Formation from the Linear to Non-Linear Regime in Polymer Nanofilms Subject to Benard-Like Instability, YU LIU, EUAN MCLEOD, SANDRA TROJAN, California Institute of Technology, 1200 E California Blvd, MC 128-95, Pasadena, CA 91125 — Researchers continue to seek novel nanopatterning techniques which unlike conventional photolithography can yield more rapid, less expensive pattern definition compatible with large areas, curved substrates and polymeric materials. Here we describe in-situ measurements of the spontaneous formation and growth of fluid elongations in molten polymer nanofilms. These films, confined in between two substrates held at different temperature, exhibit a free surface subject to thermocapillary instability. Unlike the typical Benard instability in which the fluid interface adopts an undulatory shape with small fixed amplitude, this system generates pillare-like features spaced tens of microns apart which evolve continuously toward the cooler substrate. The pillar shape and size therefore depend on the period of exposure to the thermal gradient. Optical microscopy images obtained through a sapphire window patterned with a transparent cylindrical protrusion are used to compute the pair correlation function, coordination number and Fourier transform of structure formation as a function of time. In-plane hexagonal symmetry is only observed at relatively late times and lamellar or square symmetry is all too easily obtained for tilted or curved substrates.
with measurements in bulk and in nanochannels with hydrochloric acid solutions, we verify its predictive power. Our experimental data with aqueous potassium chloride solutions in 165-nm-high silica nanochannels well, and furthermore, by comparing model predictions to independent measurements of electro-osmotic mobility and streaming current, allows us to fully characterize the slippage effect on moderate hydrodynamic slippage of the fluid at the solid-fluid interface. The effective surface charge obtained by the conductance measurements is largely higher than the commonly measured for glass surfaces. Introducing a concentration showing a saturation effect in the low concentration regime. Such effect is due to the surface correction to the channel conductance. Our model self-consistently couples chemical equilibrium models of the solvent and solute, which produce arrays of similar-like structures. For polymers with low transition temperatures, these structures solidify in place once the driving force is removed. During the actual formation process, the region in between pillars can thin substantially below 50 nm. Stabilizing van der Waals forces then become significant and can slow pillar growth and modify the instability wavelength. In this talk, we discuss results of a linear stability analysis of an initially uniform film subject to thermocapillary, capillary forces, and van der Waals forces. We then use analytic and numerical studies to explore the dynamics of film thinning in between pillar formation to better understand transitions in force balance and subsequent film deformation in non-uniform films. Results of investigations based on self-similarity and asymptotic analysis will be presented.

1 RD gratefully acknowledges support from the Caltech SURF program and the NSF.

8:39AM AP.00004 The relationship between induced fluid structure and boundary slip in nanoscale polymer films: A molecular dynamics simulation study. NIKOLAI PRIEZJEV, Michigan State University — The molecular mechanism of slip at the interface between polymer melts and weakly attractive smooth surfaces is investigated using MD simulations. In agreement with our previous studies, it is shown that the slip length passes through a local minimum at low shear rates and then increases rapidly at higher rates. We found that at sufficiently high shear rates, the slip flow over flat crystalline surfaces is anisotropic. It is demonstrated numerically that the friction coefficient (ratio of viscosity and slip length) undergoes a transition from a constant value to the power-law decay as a function of the slip velocity. The characteristic velocity of the transition correlates well with the diffusion velocity of monomers in the first fluid layer. We also show that in the linear regime, the friction coefficient is well described by a function of a single variable, which is a product of the magnitude of surface-induced peak in the structure factor and the contact density of the adjacent fluid layer. This universal relationship between the friction coefficient and induced fluid structure holds for a number of material parameters of the interface: fluid density, chain length, wall-fluid interaction energy, wall density, lattice type and orientation, thermal or solid walls. Reference: cond-mat/0007453.

8:52AM AP.00005 The Saffman-Taylor Instability Without Walls. SANDRA TROIAN, California Institute of Technology, 1200 E California Blvd, MC 128-95, Pasadena, CA — The Saffman-Taylor problem represents the unstable displacement of a more viscous fluid by a less viscous fluid under the action of an external pressure gradient. Small sinuous deformation of the separation front gives way to repeated fingering and tip-splitting. This repetitive process causes the transformation of an initially featureless front to a highly ramified curve whose fractal dimension is roughly 1.7. This instability requires that the fluids be confined by two substrates, such as in a Hele-Shaw cell, in order to force the pressure gradient which drives the flow. In this talk, we describe experiments in which this instability is observed for the first time in nanofluids freely suspended in air for which there are no confining walls. The nanofluids consist of aqueous surfactant solution containing hydrosoluble polymer. The fractal dimension $D_F$ ranges from $1 < D_F < 2$ and increases with the viscosity of the bulk solution. The expanding front appears to delineate between a surfactant-rich mobile phase and a polymer-rich less mobile phase. We discuss a phenomenological model for linearly unstable flow in which the mobility contrast incorporates both modulation in film thickness from disjoining pressure variation as well as the viscosity contrast from phase segregation. This extension generalizes our understanding of this well known phenomenon.

9:05AM AP.00006 Nanofluidics: Ionic transport through a nanochannel. ALESSandro SIRIA, LPMCN-Lyon, ANNE-LAURE BIANC, LPMCN-Lyon, CRISTOPHE YBERT, LYDERIC BOCUET, LPMCN-Lyon, LIQUID@INTERFACES TEAM — Nanopore based membranes find nowadays interesting applications such as water treatment [1] and power energy conversion [2]. We consider a single micrometric glass channel. We show how surface effect can modify the channel transport properties. We measure the ionic conductance through the channel as a function of the electrolyte concentration showing a saturation effect in the very low concentration regime. Such effect is due to the surface correction to the channel conductance [3]. The effective surface charge obtained by the conductance measurements, is largely higher than what commonly measured for glass surfaces [4]. Introducing a moderate hydrodynamic slippage of the fluid at the solid-fluid interface allows us to obtain surface charge values in agreement with what presented in literature. A quantitative agreement with independent measurement of electro-osmotic mobility and streaming current allows us to fully characterize the slippage effect on the channel electokinetic properties.

1 This work is supported by ANR program MIKADO.

9:18AM AP.00007 Hydronium-dominated ion transport in carbon-dioxide-saturated electrolytes at low salt concentrations in nanochannels. SUMITA PENNATHUR, UCSB, KRISTIAN JENSEN, JESPER KRISTENSSEN, DTU, ANDREW CRUMRINE, UCSB, MATHIAS ANDERSEN, HENRIK BRUUS, DTU — Nanochannel ion transport is known to be governed by surface charge at low ion concentrations. In this talk, we show that this surface charge is dominated by hydronium ions arising from dissolution of ambient atmospheric carbon dioxide. By refining the electrokinetic model of the nanochannel conductance for low salt concentrations, we identify a minimum conductance value before saturation at a value independent of salt concentration in the dilute limit. Our model self-consistently couples chemical equilibrium models of the silica wall and the electrolyte bulk, and is parameterized by only the surface reaction equilibrium constant for silica/hydronium reactions. The model describes our experimental data with aqueous potassium chloride solutions in 165-nm-high silica nanochannels well, and furthermore, by comparing model predictions with measurements in bulk and in nanochannels with hydrochloric acid solutions, we verify its predictive power.
larval locomotion could be analyzed in realistic, rapidly varying patterns of water movement and odor. The forces with peaks that depend on their location within the fine scale habitat topography. These data enabled us to design small-scale experiments in which concentrations encountered by larvae as they swim in the water and land on surfaces. We found that larvae have rapid on-off encounters with odors while surfaces in suitable habitats exposed to wavy, turbulent water flow. Using fouling communities and coral reefs as study systems, we measured water flow microscopic larvae that swim and respond to environmental factors (e.g. odors). Larvae are dispersed to new sites by ambient water currents and land on by turbulent ambient currents and waves in marine habitats? We addressed this question using larvae of marine invertebrates. Many benthic animals produce muscular contraction constrain the hydrodynamic efficiency of locomotion for all but a small range of sizes. Even simple forms of locomotion can be complex in which ecological release from predation apparently constrains the evolution of improved locomotory capacity, and squids, in which the fundamental limitations however, environmental and physiological reality constrains the potential for hydrodynamic innovation. We explore two heuristic examples: Antarctic scallops, and the surface reaction constant p

PENG WU, RUI QIAO, Clemson University — Electroosmotic flows (EOF) is widely used in micro/nanofluidic systems for fluid manipulation. The driving force for EOF exists only within the electrical double layers (EDLs) near charged substrates. While EOFs with thick EDLs are now well-understood, current knowledge on EOFs with EDLs thinner than a few nanometers remains limited. Specifically, experimental evidence suggests that the viscosity of interfacial fluids in EOF is higher than that of bulk fluids, but the physical origins of this universal phenomenon remain elusive. Many mechanisms such as layering of interfacial fluids, high ion concentration in EDL, and polarization of fluids in the EDL have been proposed, but a universal mechanism that encompass the breadth of experimental evidence has not been firmly established. In this work, we use molecular dynamics simulations to compute the effective viscosity of interfacial fluids in carefully controlled EOFs. We examine many mechanisms in the literature and suggest a mechanism that is capable of explaining the enhanced viscosity of interfacial fluids in EOFs regardless of the nature of the solid substrates.

9:57AM AP.00010 An investigation of nucleation-growth of bubbles using molecular dynamics simulation
TAIGA KOMATSU, The University of Tokyo, SHINICHI TSUDA, Shinsiu University, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — Microscopic phase transition phenomena have been applied in medical or industrial area recently. However, the behavior of bubble nucleation in nano-scale has not been clarified yet. In this study, a bubble nucleation-growth process was investigated in a decompressed Lennard-Jones fluid using an NVT ensemble molecular dynamics simulation. As a result, we found that cells can escape from a confined environment by swimming back through the flagellar bundle, without changing the orientation of the cell body. This maneuver involves normal-to-curly and curly-to-normal polymorphic transformations. Many phenomena will be illustrated.

Sunday, November 21, 2010 8:00AM - 10:10AM — Session AQ Mini-Symposium on Biological Perspectives on Locomotion Long Beach Convention Center 203B

8:00AM AQ.00001 E. coli swimming over agar in a thin aqueous film
HOWARD BERG, Harvard University — When cells of Escherichia coli are grown in a rich medium over somewhat soft agar (0.45%) they elongate, produce more flagella, and swarm (or flock). Their behavior is dominated by collisions: an individual cell's velocity is randomized in about 0.2 s [1]. However, cells do not swim in spirals, as they do when in a thick layer of fluid near a solid boundary [2]. This suggests that the surface of the swarm is stationary, i.e., that the cells swim in a thin film of fluid between two fixed surfaces. We showed that this is the case by following the motion of MgO smoke particles deposited at the fluid-air interface [3]. By visualizing flagella of cells in swarms, we found that cells can escape from a confined environment by swimming back through the flagellar bundle, without changing the orientation of the cell body. This maneuver involves normal-to-curly and curly-to-normal polymorphic transformations [4]. These phenomena will be illustrated.


8:26AM AQ.00002 Jet propulsion in animals: theoretical innovation and biological constraints
MARK DENNY, Stanford University — Jet propulsion is arguably the oldest and simplest form of animal locomotion, and simple hydrodynamic theory highlights the many possible ways in which animals might maximize speed and minimize metabolic cost while using jet propulsion to travel from one point to another. However, environmental and physiological reality constrains the potential for hydrodynamic innovation. We explore two heuristic examples: Antarctic scallops, in which ecological release from predation apparently constrains the evolution of improved locomotory capacity, and squids, in which the fundamental limitations of muscular contraction constrain the hydrodynamic efficiency of locomotion for all but a small range of sizes. Even simple forms of locomotion can be complex in a biological context.

8:52AM AQ.00003 Locomotion by microscopic organisms in turbulent ambient water flow
M.A.R. KOEHL, University of California Berkeley — How is the locomotion of microscopic organisms swimming or moving across the substratum affected by turbulent ambient currents and waves in marine habitats? We addressed this question using larvae of marine invertebrates. Many benthic animals produce microscopic larvae that swim and respond to environmental factors (e.g. odors). Larvae are dispersed to new sites by ambient water currents and land on surfaces in suitable habitats exposed to wavy, turbulent water flow. Using fouling communities and coral reefs as study systems, we measured water flow across them in the field and recreated it in flumes where we could quantify on the scale experienced by larvae (mm’s, ms’s) the instantaneous water velocities and concentrations of odors released from surfaces where larvae settle. We used these data to determine the temporal patterns of water velocities and odor concentrations encountered by larvae as they swim in the water and land on surfaces. We found that larvae have rapid on-off encounters with odors while swimming through fine filaments of odor swirling in unscented water, experience varying shear, and after landing are exposed to rapidly fluctuating hydrodynamic forces with peaks that depend on their location within the fine scale habitat topography. These data enabled us to design small-scale experiments in which larval locomotion could be analyzed in realistic, rapidly varying patterns of water movement and odor.
are shown as plots of dimensionless volume difference between the two droplets against electrical Bond number (ratio of electric to surface tension force). These...
9:05AM AR.00006 Electrodeless electro-hydrodynamic gentle printing of personalized medicines. BORIS KHUSID, EZINWA ELELE, YUEYANG SHEN, New Jersey Institute of Technology — Drop-on-demand (DOD) principle appears to be a particular promising approach for manufacturing personalized treatments carefully tailored to a patient’s genetic background. The authors have recently developed a DOD method for gentle printing of personalized medicines. A fluid is infused into an electrically insulating nozzle to form a pendant drop. A sufficiently strong voltage pulse is applied to external electrodes to stretch the pendant drop until it touches an electrically insulating film and forms a liquid bridge. As the liquid bridge is intentionally formed in an unstable configuration, it breaks up, creating two drops, one on the film and the other hanging from the nozzle. To prove the validity and versatility of the method, experiments are conducted on fluids whose viscosity, conductivity, dielectric constant, and surface tension vary over a broad range, respectively: 1-1045 cp, 0.02-290 μS/cm, 9-78, and 41-72 dyn/cm. We present a scaling analysis that captures the essential physics of drop evolution and provides the critical design guidelines. The work was supported by NSF Engineering Research Center on Structured Organic Particulate Systems.

9:18AM AR.00007 Effect of Nozzle shape on droplet generation stability in EHD Inkjet. JIYOUNG KIM, VU DAT NGUYEN, JIHOON KIM, SI BUI QUANG TRAN, DOYOUNG BYUN, Konkuk University — This study reports an effect of nozzle shape that offers better uniformity and stable operation in jetting performance. The stability of a liquid meniscus is important for the ability to eject a small liquid droplet. We investigated jetting performance in term of uniformity of patterns for different nozzle shapes in DC based EHD inkjet. To generate droplet with nozzles which have different shape, two types of glass capillaries are used in this study; one is a circular capillary and the other is a square capillary. The square edges are shown to keep the liquid inbound better. When the liquid is supplied to the circular nozzle, it grows and limits by the outer edge of the nozzle. However, in the case of the square nozzle, the meniscus can be sustained stably by the square edges. The ejections were recorded with a high speed camera and analysed to examine the difference in dynamic movement of meniscus. The repeatability of jetting is more periodic in the case of the square nozzle.

9:31AM AR.00008 A Hybrid Inkjet Printer Utilizing Electrohydrodynamic Jetting and Piezoelectric Actuation. VU DAT NGUYEN, DOYOUNG BYUN, Konkuk University — This research demonstrates a hybrid electrohydrodynamic (EH) inkjet printing technique that offers better uniformity and stable operation in drop-on-demand (DOD) patterns compared to the conventional methods. This hybrid technique takes advantage of both electrohydrodynamic and piezoelectric methods where a piezoelectric actuator is used to supply a fixed volume of ink to the nozzle’s exit for every jetting period, and the electrohydrodynamic technique is used to form ink droplets. Experimental results show that the pattern uniformity improves significantly when ink was supplied to the nozzle exit at a controlled rate using piezoelectric actuation. This hybrid technique can be applied to small scale nozzle to obtain high resolution printing.

9:44AM AR.00009 Flight Behavior of a Charged Droplet in Electrohydrodynamics (EHD) Inkjet printing using DC and AC signal. HADI TEGUH YUDISTIRA, VU DAT NGUYEN, DOYOUNG BYUN, Department of Aerospace Information Engineering, Konkuk University, Seoul, Republic of Korea, NANO/MICRO SYSTEM LABORATORY KONKUK UNIVERSITY TEAM — Flight behaviors of charged droplets such as reflection, deflection, and retreat, are presented for electrohydrodynamic (EHD) inkjet printing. Experimental results show that the flight paths of charged droplets may deviate from their regular straight route, i.e., directly from the nozzle to the substrate. Depending on the droplet charge and applied electric field, droplets may deflect and reflect on a substrate, or retreat back to the meniscus. The retreat phenomenon is one of the behaviors of the charged droplet due to interactions between a droplet, meniscus, and a substrate. The droplet reversely moves back to the meniscus due to loss of charges after the second fission. To estimate the amount of charge on both the droplet and the meniscus, the Rayleigh limit was used.

9:57AM AR.00010 Oscillations of an asymmetric double droplet system. SANTHOSH RAMALINGAM, OSMAN BASARAN, Purdue University — When a small cylindrical hole in a plate is overfilled with a liquid, a double droplet system (DDS) is created, consisting of a sessile (top) drop and a pendant (bottom) drop. For small hole radii, R, equilibrium shapes of both drops are sections of spheres. Due to the drops’ spherical surfaces and its miniature size, a DDS serves as well as a micro lens and a DDS oscillating about its equilibrium shape can be used as a fast focusing lens. In this talk, we consider a DDS consisting of an isothermal, incompressible Newtonian liquid of constant density and constant viscosity that is surrounded by a gas and where the air-liquid surface tension is constant. Exciting the DDS by oscillating in time (a) the pressure in the gas surrounding either drop (pressure excitation), (b) the plate perpendicular to its plane (axial excitation), and (c) the hole radius (radial excitation), the natural modes of oscillation are identified from resonances during frequency sweeps. Here, we study numerically, using the Galerkin finite element method, the oscillation modes of a DDS in which the combined volume of the pendant and sessile drops is greater than the critical volume corresponding to that of a sphere of radius R. The frequencies are shown to accord well with experimental observations and, in the limit of vanishing plate thickness and negligible viscous effects, the mode shapes and frequencies are also shown to agree with theoretical predictions.

Sunday, November 21, 2010 8:00AM - 10:10AM – Session AU Multiphase Flows I Hyatt Regency Long Beach Regency A

8:00AM AU.00001 Effect of microstructural anisotropy on the fluid-particle drag force. WILLIAM HOLLOWAY, JIN SUN, SANKARAN SUNDARESAN, Princeton University — The permeabilities of particle assemblies with anisotropic microstructures have been determined through lattice-Boltzmann simulations. Such assemblies were created by subjecting them to simple shear in a periodic domain. The extent of anisotropy depends on the scaled rate of deformation of the particle assembly |D|/√T, where T is the particle diameter, |D| is the magnitude of the rate of strain tensor, and T is the granular temperature. The anisotropy of the permeability tensor increases with the scaled rate of deformation and the particle volume fraction, and it can readily be rationalized in terms of the structure tensor of the assembly. A model for the anisotropic permeability is proposed in terms of mean free path of the deformed assembly, and the rate of strain tensor.
We find that acceleration PDFs show stretched exponential tails, the shape being independent of Re. Microbubbles of size comparable to Kolmogorov’s lengthscale are injected in a turbulent water channel. By varying the Reynolds number, the simulation matches all parameters of an experiment by Sanders, et al. (J. Fluid Mech., 2006). The bubbly suspension is dilute and one-way coupled equations are used. The temporal evolution of the bubble dispersion, probability density functions of the forces on a bubble and void–fraction profiles will be presented, and the impact of bubble behavior on drag reduction and the effect of cavitation number will be discussed.

8:26AM AU.00003 Lagrangian statistics of bubbles in a turbulent boundary layer†. MICHAEL MATTSON, KRISHNAN MAHESH, University of Minnesota, Twin Cities — We are developing the capability to simulate bubbly flows in complex geometries using unstructured grids and an Euler–Lagrangian methodology. In the Lagrangian bubble model, the bubbles are treated as a dispersed phase in the carrier fluid, and individual bubbles are point–particles governed by an equation for bubble motion. The behavior of the bubble radius is determined by integrating the Rayleigh–Plüss equation. For this talk, direct numerical simulation is used to solve the Navier–Stokes equations for a spatially–evolving turbulent boundary layer (Re_{\text{eq}} = 600 – 1800) and bubbles are injected into the near–wall region. Except for the Reynolds number, the simulation matches all parameters of an experiment by Sanders, et al. (J. Fluid Mech., 2006). The bubbly suspension is dilute and one–way coupled equations are used. The temporal evolution of the bubble dispersion, probability density functions of the forces on a bubble and void–fraction profiles will be presented, and the impact of bubble behavior on drag reduction and the effect of cavitation number will be discussed.

†Supported by the U.S. Office of Naval Research under ONR Grant N00014-07-1-0420.

8:39AM AU.00004 Interfacial Force Model Development for Turbulent Bubbly Flows, DILLON SHAVER, IGOR BOLOTMOV, STEVEN ANTAL, MICHAEL PODOWSKI, Rensselaer Polytechnic Institute — Typically, a Reynolds averaged Navier-Stokes (RANS) simulation of bubbly flows makes use of interfacial force models which represent the interaction between the bubbles and the continuous phase. The modeled forces include drag, virtual mass, turbulent dispersion, and lift. A direct numerical simulation (DNS) fully resolves turbulent fluctuations in velocity and, when coupled with the level set method, can simulate a two-phase flow without relying on interfacial force models. Results from DNS can provide a level of insight into flow characteristics not easily achievable with traditional experimental methods. This makes DNS ideal for developing interfacial force models for use with RANS codes. Turbulent, air/water, bubbly flows in a channel have been previously simulated using the DNS code, PHASTA. Utilizing the time–averaging concept, average velocities of the two phases, void fraction, turbulent kinetic energy, and turbulence dissipation rate distributions are calculated from the DNS data. This information is then used to develop and calibrate the interfacial force models used in the RANS code, NPHASE-CMFD. Two cases are analyzed. The first is of many small, spherical bubbles of 0.9 mm diameter. The other is of a single, large, cap bubble of 3.625 mm equivalent diameter. Both simulations correspond to the liquid Reynolds number of 11,200, based on the hydraulic diameter.

8:52AM AU.00005 Light Particles in Turbulence: acceleration statistics, JULIAN MARTINEZ MERCADO, VIVEK NAGENDRA PRAKASH, YOSHIYUKI TAGAWA, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, University of Twente — Three-dimensional Lagrangian Particle Tracking experiments are used to study acceleration statistics of light particles (3 \rho_p/(\rho_\text{fluid} + 2 \rho_p) = 3) in isotropic turbulence. Microbubbles of size comparable to Kolmogorov’s lengthscale are injected in a turbulent water channel. By varying Re we study the effect of changing the turbulent lengthscale on the statistics for a fixed particle size. We compare our results with previous experimental and numerical data on particles in turbulence. We find that acceleration PDFs show stretched exponential tails, the shape being independent of Re. The acceleration autocorrelation shows that light particles decorrelate faster than tracer or heavy particles. The correlation drops rapidly to zero in less than one Kolmogorov’s timescale. The decorrelation time increases with Re. This trend is in agreement with previous experimental data for different flows and with numerical simulations.

9:05AM AU.00006 Light particles in turbulence: velocity statistics, JULIAN MARTINEZ MERCADO, YOSHIYUKI TAGAWA, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, University of Twente — We conduct experiments to study light particles in turbulence using Particle Tracking Velocimetry (PTV) in three-dimensions. Microbubbles are dispersed in a homogenous and isotropic turbulent flow in the Twente water tunnel. The size of the microbubbles is fixed and is comparable to the Kolmogorov length scale of the flow. The Lagrangian velocity statistics of the microbubbles are obtained from the trajectories captured using PTV. The velocity statistics (PDF, autocorrelation and structure functions) of microbubbles are studied at different Re and compared with previous experiments and numerics for particles in turbulence. The velocity PDF of the 3 velocity components (x, y and z) show a robust gaussian profile (independent of Re) with flatness values between 2.74 to 3.25. We calculate the velocity autocorrelation and find that the decorrelation time increases with increasing Re. We also calculate the second and fourth - order velocity structure functions and find a reasonable agreement with previous numerical simulations.

9:18AM AU.00007 Local bubble distribution in bubbly turbulent Taylor-Couette flow, DANIELA NAREZO, DENNIS VAN GILS, CHAO SUN, DETLEF LOHSE, University of Twente — In turbulent Taylor-Couette flow, the injection of bubbles reduces the global drag on the cylinder surfaces. The previous bubbly turbulent drag reduction measurements in TC flow were mainly based on the global torque, which is not sufficient to understand the mechanism of bubbly drag reduction. One of the key issues is the actual bubble distribution inside the TC gap. Using optical fibers placed inside the TC gap, we scanned the local bubble distribution in the radial direction. An extension of this technique is a four-point optical fiber probe, which enables to retrieve the bubble velocity vector and aspect ratio.

9:31AM AU.00008 An arbitrary-Lagrangian-Eulerian method for simulating particle-bubble interactions, TONG QIN, PENGTAO YUE, SAAD RAGAB, Virginia Tech — Particle-bubble interaction is an important process in flotation. This problem is difficult in that it involves three phases: solid particles, gas bubbles, and surrounding liquid. In this work, an arbitrary-Lagrangian-Eulerian (ALE) approach is developed for the direct numerical simulation. A moving triangular mesh is used to track the surfaces of rigid solid particles and deformable gas bubbles. The gas motion inside each bubble is neglected, and the pressure is determined by the isothermal gas law. The equations for the particle motion and the Navier-Stokes equations for the liquid motion are solved in a unified finite element framework. The whole system is solved by an implicit scheme which is second order in time. In the end, we will show results on the head-on collision between a bubble and a particle and the subsequent film drainage process. Depending on the collision conditions, the particle may attach to the bubble or be bounced back. Comparisons with experiments will also be presented.
A model for particle density based on a Cahn-Hilliard formalism. We investigate Landau-Levich coating of a solid wall by filtration. Just in time, Anette Hosoi, Massachusetts Institute of Technology — We investigate Landau-Levich coating of a solid wall by filtration.

In almost all standard Taylor-dispersion analysis, thermal fluctuations (Brownian motion) are assumed to be the underlying stochastic element driving particles at single particle resolution within a large (caused by depletion interactions) and an elastic component to the flow. The sedimentation dynamics are governed by the formation, sedimentation and sedimentation of colloidal particles in polymer supplemented solution. The polymers enrich the dynamics of sedimentation by adding both particle attraction and repulsion.

We study the sedimentation of colloidal particles in polymer supplemented solution. The polymers enrich the dynamics of sedimentation by adding both particle attraction (caused by depletion interactions) and an elastic component to the flow. The sedimentation dynamics are governed by the formation, sedimentation and consequent breakup of poroelastic clusters of many particles. By making use of a custom built laser sheet microscope we are able to track Brownian one-micron particles at single particle resolution within a large (~cm sized) cell. This way we can resolve between bulk and boundary effects.

Partial Support from National Science Council of Taiwan

Inertial particles lead to highly non-Gaussian distributions. In the context of moment methods for solving the kinetic equation, the latter can be successfully modeled using quadrature-based moment methods. In this work, such methods are extended to allow for a continuous transition between a delta-function representation of the quadrature nodes and a multi-Gaussian representation. Examples with 1D and 2D velocity phase spaces are presented to illustrate the advantages of multi-Gaussian quadratures. In particular, the representation of the spatial fluxes is considerably improved for cases where the velocity distribution is close to Gaussian.

Partial Support from National Science Council of Taiwan
9:05AM AV.00006 Particle-laden thin film flows on an incline: experiments and equilibrium theory, JACOB BOURICIIUS, Mathematics, Harvey Mudd College, TRYSTAN KOCH, Physics, Harvey Mudd College, PAUL LATTERMAN, Bioengineering, UCLA, BRIAN LE, Physics, UCLA, SAMANTHA MESURO, Mathematics, Harvey Mudd College, NEBOJSA MURISIC, ANDREA BERTOZZI, Mathematics, UCLA — We focus on particle-laden thin film flows on an incline. Experiments are carried out where inclination angle, bulk particle volume fraction, liquid viscosity, and particle size are varied. We classify experimental runs based on observed settling regime: settled, where particle settle out of the flow and fingering instability develops at the front (low angles and concentrations); ridged (high angles and concentrations); and well-mixed (intermediate values). We also uncover the transient nature of the well-mixed regime. In addition, in our experiments, the suspension and particulate front motion is tracked using a camera/laser set-up. Using image processing, we are able to extract the instantaneous thin film profiles and record the front motion for all observed flow regimes. The theoretical model we consider is based on equilibrium theory and it balances hindered settling of particles due to gravity against shear-induced migration. Finally, the predictions of this model are shown to be in excellent agreement with our experimental data for settling.

9:18AM AV.00007 A dynamic model for particle-laden thin film flows on an incline, NEBOJSA MURISIC, ANDREA BERTOZZI, Department of Mathematics, UCLA — Particle-laden flows are important in a variety of contexts, where transport and manipulation of suspensions occur. We consider gravity driven flows of particle-laden thin films on an incline. In the experiments, three distinct regimes are observed: particles either settle out of the flow (low bulk particle volume fraction and inclination angle), aggregate at the moving front (high bulk particle volume fraction and inclination angle), or remain well-mixed (intermediate values). Our dynamic model relies on the suspension and particle fluxes resulting from previously derived equilibrium model, where shear-induced migration balanced hindered settling due to gravity. The dynamics is modeled using a system of two scalar hyperbolic conservation laws, describing suspension and particulate front motion. We proceed by discussing a few aspects of the rich mathematical structure of these laws and their physical interpretation. Finally, the governing system is solved numerically, and simulation results are shown to agree well with the experimental data regarding front propagation and settling mode.

9:31AM AV.00008 Cross-stream migration of compliant particles in microfluidic channels, ALEX KILIMNIK, Georgia Institute of Technology, SOOJUNG CLAIRE NUR, DINO DI CARLO, UCLA, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using a 3D hybrid lattice Boltzmann and lattice spring computational method, the motion of rigid and soft particles in a pressure-driven microfluidic flow was examined. The particles were modeled as neutrally buoyant fluid-filled elastic shells. The equilibrium positions of these particles were obtained in a low-Reynolds-number flow while accounting for non-linear inertial effects. Microchannels of different width were examined and it was found that the equilibrium position of the rigid particles moves away from the channel walls as the ratio between particle diameter and channel width increases. Furthermore, it was found that capsule deformability enhances the particle migration toward the channel centerline. The simulation results were compared with experimental data obtained with varying size and viscosity oil droplets suspended in water indicating favorable agreement. These findings could aid in the design of devices to sort particles based on their mechanical stiffness. 

1This research was supported in part by the NSF (CBET-0930501) and through the NSF TeraGrid computational resources.

9:44AM AV.00009 Rotation and Alignment of Rods in Two-Dimensional Chaotic Flow, JERRY GOLLUB, JEFFREY GUSTO, MONICA KISHORE, Haverford Coll., SHIMA PARSA MOGHADDAM, GREG VOTH, Wesleyan U., NICHOLAS OUELLETTE, Yale U. — We study the dynamics of rod shaped particles in two-dimensional electromagnetically driven fluid flows. Two separate flows are compared: one with time-periodic flow and the other with non-periodic flow. Video particle tracking is used to make accurate measurements of the motion and orientation of rods along with the carrier fluid velocity field. Measured rod rotation rates are in agreement with predictions for ellipsoidal particles based on the measured velocity gradients at the center of the rods. There is little dependence on length for the rods we studied (up to 53% of the length scale of the forcing). Rods are found to align weakly with the direction of extensional strain. However, the alignment is much stronger with the direction of Lagrangian stretching defined by the eigenvectors of the Cauchy-Green strain tensor. A simple model of the stretching process predicts the degree of alignment of rods with the stretching direction.

1 NSF Grants DMR-0547712 and DMR 0803153

9:57AM AV.00010 Measuring the rotation rate of rod-shaped particles in 3D turbulence, SHIMA PARSA, Wesleyan University, NICHOLAS T. OUELLETTE, Yale University, GREG A. VOTH, Wesleyan University — We study the rotation of rod-shaped particles in turbulent flow using stereoscopic particle tracking with high speed video. With images from multiple cameras we are able to extract the position and orientation of rods as a function of time in a flow between oscillating grids. We work in the low density limit where rod-rod interactions can be ignored. Rod rotation is determined by the local velocity gradient so measurement of the rotation rate variance provides an indirect way to access the second moment of velocity gradient of the fluid. Development of methods for tracking rods in turbulence shows great promise both for understanding many flows containing inertial rods and as a means of extracting small scale properties of the flow along Lagrangian trajectories.

Sunday, November 21, 2010 8:00AM - 10:10AM – Session AW Instability: Richtmyer-Meshkov/Rayleigh-Taylor I Hyatt Regency Long Beach Regency C

8:00AM AW.00001 Richardson effects in turbulent buoyant flows, RENAUD BIGGI, Ecole Normale Superieure, Cachan, France, GUILLAUME BLANQUART, Caltech, CA, USA — Rayleigh Taylor instabilities are found in a wide range of scientific fields from supernova explosions to underwater hot plumes. The turbulent flow is affected by the presence of buoyancy forces and may not follow the Kolmogorov theory anymore. The objective of the present work is to analyze the complex interactions between turbulence and buoyancy. Towards that goal, simulations have been performed with a high order, conservative, low Mach number code [Desjardins et. al. JCP 2010]. The configuration corresponds to a cubic box initially filled with homogeneous isotropic turbulence with heavy fluid on top and light gas at the bottom. The initial turbulent field was forced using linear forcing up to a Reynolds number of Re_L = 55 [Meneveau & Rosales, POF 2005]. The Richardson number based on the rms velocity and the integral length scale was varied from 0.1 to 10 to investigate cases with weak and strong buoyancy. Cases with gravity as a stabilizer of turbulence (gravity pointing up) were also considered. The evolution of the turbulent kinetic energy and the total kinetic energy was analyzed and a simple phenomenological model was proposed. Finally, the energy spectra and the isotropy of the flow were also investigated.
8:13AM AW.00002 Wavelet-Based Simulations of Rayleigh-Taylor Instability. SCOTT RECKINGER, University of Colorado, DANIEL LIVESCU, Los Alamos National Laboratory, OLEG VASILYEV, University of Colorado — The Rayleigh-Taylor instability is investigated using numerical simulations on an adaptive mesh, performed with the Adaptive Wavelet Collocation Method (AWCM). The wide range of scales present in the development of the instability are efficiently resolved with AWCM, due to the physics-based adaptivity and direct error control of the method. The problem is initialized consistent with the solutions from linear stability theory, where the base state is the diffusive mixing of incompressible variable density fluids. Of interest are the variable density and compressibility effects on the departure time from the linear growth, the onset of strong non-linear interactions, and the late-time behavior of the fluid structures. Simulations performed for a single-mode perturbation in the incompressible limit match the early time linear growth, the terminal bubble velocity, and a reacceleration region. In order to investigate the turbulent mixing rates of the pure heavy and light fluids within the Rayleigh-Taylor mixing layer, simulations of compressible homogeneous isotropic turbulent mixing in a triply-periodic domain are also performed.

8:26AM AW.00003 Statistical model for turbulent transition by variable-density pressure-gradient-driven mixing. J. BAKOSI, J.R. RISTORCELLI, Los Alamos National Laboratory — A Monte-Carlo method for variable-density (VD) pressure-gradient-driven turbulence has been developed. VD effects due to non-uniform mass concentrations (e.g. mixing of different-density species) are considered. The model numerically computes the full time-evolution of the joint probability density function (PDF) of fluid density and velocity in a non-stationary Rayleigh-Taylor flow, that develops from quiescent state to a laminar stage, through transition to fully developed turbulence and dissipative decay. The coupled model for hydrodynamics and mixing is designed for arbitrary Atwood numbers. The main characteristics of the method are: (1) It eliminates the need for quasi-equilibrium assumptions, gradient diffusion hypotheses, modeling of the mass flux and of the density-specific-volume covariance; (2) The mixing state is represented by the density PDF; (3) It captures the density skewness, due to large differential accelerations of different-density species; and (4) It represents both small and large scale anisotropy.

8:39AM AW.00004 Direct numerical simulations of Rayleigh-Taylor instability with gravity reversal. MARK PETERSEN, DANIEL LIVESCU, ROBERT GORE, Los Alamos National Laboratory — We have conducted high resolution, high Reynolds number Direct Numerical Simulations (DNS) of the Rayleigh-Taylor (RT) instability on the 0.5 petaflop, 150k compute cores BG/L. Dawn supercomputer at Lawrence Livermore National Lab. This includes a suite of simulations with Atwood number ranging from 0.04 to 0.9 and grid size of 1024^3 by 4096, and a high resolution simulation of grid size 4096^3 and Atwood number of 0.75. After the layer width has developed substantially, additional branched simulations have been run under reverse gravity and zero gravity conditions. The simulations provide an extensive database to study Rayleigh-Taylor turbulence, including mixing layer growth rate and self-similar behavior, turbulence and mixing asymmetries, and spectral characteristics. Individual terms in the moments transport equations are recorded to develop and validate turbulence closure models.

8:52AM AW.00005 Turbulence characteristics in the variable-density Rayleigh-Taylor mixing layer. DANIEL LIVESCU, MARK PETERSEN, ROB GORE, Los Alamos National Laboratory — The turbulence generated in the Rayleigh-Taylor mixing layer is studied using data from Direct Numerical Simulations on up to 4096^3 meshes. The simulations cover the range of Atwood numbers A = 0.04 – 0.9 in order to study small departures from the Boussinesq approximation as well as large Atwood number effects. The results show that, although the layer width becomes self-similar relatively fast, the lower order terms in the self-similar expressions for turbulence moments have long-lasting effects and derived quantities, such as the turbulent Reynolds number, are slow to follow the self-similar predictions. This has important consequences for moment closures, which generally assume full, asymptotic self-similarity. The results also show that at large Atwood numbers, the turbulence structure changes qualitatively and various turbulence moments become asymmetric. These asymmetries, together with that of the mixing itself, have a profound influence on the shape of the mixing layer.

9:05AM AW.00006 Revised Froude number for Rayleigh-Taylor flow with secondary instabilities. KARTHIK MAHADEVAN MUTHURAMAN, PRAVEEN RAMAPRABHU, UNC Charlotte, GUY DIMONTE, Los Alamos National Laboratory, PAUL WOODWARD, University of Minnesota, CHRIS FRYER, GABE ROCKEFELLER, Los Alamos National Laboratory, YUAN-NAN YOUNG, NJIT — Recent simulations [1] and experiments [2] have shown the late-time Rayleigh-Taylor (RT) saturation velocity is sensitive to the appearance of secondary Kelvin-Helmholtz (KH) vortices. Specifically, RT bubbles experience a late surge due to the induced velocity of the KH vortices and saturate at a Froude number twice that predicted by potential flow models [3]. We describe this picture with a simple toy model that idealizes the KH rollups as a pair of counter-rotating point vortices. From classical linear theory, the KH growth rates depend on several parameters such as viscosity, surface tension, and density difference between the fluid streams. We have studied the influence of these parameters on the fundamental RT mode using high aspect ratio, single mode numerical simulations, and will discuss our findings. At very late time, turbulent mixing occurs due to further instabilities. The results are expected to be of relevance to turbulent mix models that are based on bubble growth and merger. [1] Ramaprabhu, P. et al. 2006, Physical Review E. 74, 066308. [2] Wilkinson, J.P. & Jacobs, J.W. 2007, Phys. Fluids 19, 124102. [3] Goncharov, V.N. 2002 Physical Review Letters 88, 134502.

9:18AM AW.00007 Non-linear effects in the combined Rayleigh-Taylor and Kelvin-Helmholtz Instabilities. BRITTON OLSON, JOHAN LARSSON, SANJIVA LELE, Stanford University — The combined Rayleigh-Taylor (RT) and Kelvin-Helmholtz (KH) instability has been studied extensively in the linear regime. We have performed studies outside the linear regime by means of Direct Numerical Simulation (DNS) and Large-Eddy Simulation (LES) where relatively little attention has been devoted. Motivation for research in this area has traditionally been plasma physics applications such as Inertial Confinement Fusion (ICF) and Type-1a supernovae collapse. Results of linear stability analysis for a discontinuous interface which combines RT with KH show that for all parameters defining the instabilities, shear addition will increase the growth rate of the RT instability. Our results show that outside this linear regime, shear in fact does mitigate the spreading rate of the RT mixing region. We present a physical explanation of this phenomenon and simple scaling laws which provide a collapse of the data. We further provide a method for determining the optimal amount of velocity shear that will effectively minimize the early time peak mixing rate.

1This work was made possible by funding from the LDRD program at Los Alamos National Laboratory through project numbers 20070195ER and 20090058DR.

This work was supported by DOE-SciDAC2 (Grant DE-FC02-06-ER25787) and the Computational Science Graduate Fellowship (CSGF).
9:31AM AW.00008 Self-Similar Solutions of Reynolds-Averaged Navier-Stokes Models for Rayleigh–Taylor Instability-Induced Turbulence and Mixing1. OLEG SCHILLING, Lawrence Livermore National Laboratory — Many applications in which modeling the effects of mixing induced by interfacial hydrodynamic instabilities is important require accurate transport closures for fusion and astrophysics, require a Reynolds-averaged Navier-Stokes (RANS) description due to the prohibitively large range of scales present. In appropriate limits, the RANS equations typically admit self-similar solutions that are useful for developing insights into the late-time behavior of turbulence and mixing. In addition, these solutions provide constraints on model coefficients through large-scale observables, other constraints through coefficient relationships, expressions for closures of the transport equations, and checks on numerical solutions of the full RANS equations. Analytical and semi-analytical solutions of two- and four-equation RANS models that include descriptions of scalar turbulence, are derived in various limits for Rayleigh-Taylor instability. The implications of the coefficient constraints on closure modeling of terms in the RANS model are discussed. The results of this study are also related to state-of-the-art simulations and experiments.

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

9:44AM AW.00009 A Posteriori Tests of a Three- and Four-Equation Advanced Reynolds–Averaged Navier–Stokes Model for Rayleigh–Taylor Turbulent Mixing1, GREGORY BURTON, OLEG SCHILLING, Lawrence Livermore National Laboratory — A high-order, multicomponent implementation of a three- and four-equation, variable-density incompressible Reynolds-averaged Navier-Stokes model incorporating both mechanical and scalar turbulence is used to simulate Rayleigh-Taylor turbulent mixing with an Atwood number equal to one-half. The closures in this model were previously tested a priori against the large Reynolds number compressible Reynolds-averaged Navier-Stokes model incorporating both mechanical and scalar turbulence is used to simulate Rayleigh-Taylor turbulent mixing. The implications of these results for advanced modeling of Rayleigh-Taylor turbulent mixing are discussed.

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

9:57AM AW.00010 Comparisons of a Reynolds-Averaged Navier–Stokes Model with Self-Similar Solutions for Large Atwood Number Rayleigh–Taylor Mixing1, RHYS ULERICH, University of Texas at Austin, OLEG SCHILLING, Lawrence Livermore National Laboratory — A new high-order, multicomponent, weighted essentially nonoscillatory (WENO) implementation of a three- and four-equation Reynolds-averaged Navier-Stokes (RANS) model incorporating both mechanical and scalar turbulence is used to simulate intermediate-to-large Atwood number Rayleigh-Taylor turbulent mixing. The predicted RANS mixing layer evolution is compared with the analytical self-similar solutions of the transport equations. The terms in the transport equation budgets are compared in detail to their profiles across the mixing layer predicted by the DNS. The implications of these results for advanced modeling of Rayleigh-Taylor turbulent mixing are discussed.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344 and by the DOE National Nuclear Security Administration under Award Number DE-FC52-08NA286.

Sunday, November 21, 2010 8:00AM - 10:10AM —
Session AX Aerodynamics I: Flapping Wings

8:00AM AX.00001 Aerodynamics of high frequency flapping wings1, ZHENG HU, JESSE ROLL, BO CHENG, XINYAN DENG — We investigated the aerodynamic performance of high frequency flapping wings using a 2.5 gram robotic insect mechanism developed in our lab. The mechanism flaps up to 65Hz with a pair of man-made wing mounted with 10cm wingtip-to-wingtip span. The mean aerodynamic lift force was measured by a lever platform, and the flow velocity and vorticity were measured using a stereo DPIV system in the frontal, parasagittal, and horizontal planes. Both near field (leading edge vortex) and far field flow (induced flow) were measured with instantaneous and phase-averaged results. Systematic experiments were performed on the man-made wings, cicada and hawk moth wings due to their similar size, frequency and Reynolds number. For insect wings, we used both dry and freshly-cut wings. The aerodynamic force increase with flapping frequency and the man-made wing generates more than 4 grams of lift at 35Hz with 3 volt input. Here we present the experimental results and the major differences in their aerodynamic performances.

1This work was supported by NSF Grant 0545931.

8:13AM AX.00002 Aerodynamic damping during body translation in animal flight: modeling and experimental results of flapping counter force (FCF)1, BO CHENG, ZHENG HU, XINYAN DENG — Body movements of flying animals change their effective wing kinematics and influence aerodynamic forces. Our previous studies found that substantial aerodynamic damping was produced by flapping wings during body rotation through a passive mechanism we termed flapping counter-torque (FCT). Here we present the aerodynamic force modulation in MAVs.

1This work was supported by NSF Grant 0545931.

8:26AM AX.00003 Aerodynamics of flapping wings with fluttering trailing edges, LIANG ZHAO, ZHENG HU, JESSE ROLL, XINYAN DENG — Our previous work on the aerodynamics of passive flexible flapping wings showed that there is a strong relationship between the dynamics of trailing edge and the size of the leading edge vortex, therefore aerodynamic forces. Here we investigated the aerodynamic effects of active trailing edges. The experiments were conducted on a model flapping wing in an oil tank. During static tests, the trailing edge bending angle was held constant from the angle of attack of the upper portion of the rigid wing. For dynamic cases, the trailing edge was controlled to flutter with a prescribed frequency and amplitude. Force measurements and PIV results showed that trailing edge flexion/camber strongly correlates with the leading edge vortex and the aerodynamic forces. In addition, large instantaneous force variations are observed in the dynamic flapping cases, suggesting that trailing edge can be used for force modulation in MAVs.
The effects of the wing platform on the aerodynamics performance of finite-span flapping wings were elucidated in the terms of the evolutions and dynamic criteria in order to demonstrate the structures formed around the wing propagates into the wake.

The stability condition of the flag was obtained and compared to the recent theoretical models and numerical simulations. Afterwards, the nonlinear dynamics of different sizes and flexural rigidities were used. Image processing technique was used and the time series of a given point on the edge of the flag was analyzed. 

Investigate experimentally the phenomenon of the flapping of a flag, placed within a low turbulent axial flow inside a small scale wind tunnel test section. Flags of different sizes and flexural rigidities were used. Image processing technique was used and the time series of a given point on the edge of the flag was analyzed. 

The stability condition of the flag was obtained and compared to the recent theoretical models and numerical simulations. Afterwards, the nonlinear dynamics of different sizes and flexural rigidities were used. Image processing technique was used and the time series of a given point on the edge of the flag was analyzed.
We thank CNPq and CAPES (Brazilian Agencies) for financial support.

1 EPSRC Grant number EP/E027393

8:26AM AY.00003 Breakup of an electrified viscous thread with charged surfactants1, DEVIN CONROY, RICHARD CRASTER, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — The dynamics and breakup of electrified viscous jets in the presence of ionic insoluble surfactants are investigated. Axisymmetric configurations are considered and the jet is surrounded by a concentrically placed cylindrical electrode with a constant voltage potential. The annular region between the jet and the electrode is taken to be inviscid and an electric field is set up there and drives the flow, along with other physical mechanisms including capillary instability and viscous effects. The jet fluid is taken to be a symmetric electrolyte and modeling of the cationic and ionic species is used by consideration of the Nernst-Planck equations in order to find the volume charge density that influences the electric field in the jet. A positively charged insoluble surfactant is present at the interface and its evolution as well as the resulting value of the local surface tension coefficient, are coupled to the voltage potential at the interface. The resulting coupled nonlinear systems are derived using a slender jet approximation. We show the jet ruptures in finite time provided the outer electrode is sufficiently far away, and demonstrate how the dimensionless parameters can be used to control the size of the satellite drops and time to breakup. Pinching solutions follow the self-similar dynamics of clean viscous jets at times close to the breakup time.

1 EPSRC Grant number EP/E056466

8:39AM AY.00004 The Dynamics of Drop Impact, JOHN KOLINSKI, Harvard School of Engineering and Applied Sciences, SHMUEL M. RUBINSTEIN, Department of Physics and Harvard School of Engineering and Applied Sciences, SHREYAS MANDRE, Brown University, L. MAHADEVAN, Harvard School of Engineering and Applied Sciences — There are many aspects of the dynamics of liquids wetting solid surfaces that are not fully understood. One such aspect is what happens at the first instance of contact. We study the dynamics of a partially wetting fluid drop as it approaches a solid surface with velocities ranging from microns- to meters-per-second. We use TIR (total internal reflection) microscopy to probe what happens immediately after the first contact. The drop impacts the solid surface and the resulting deformation process is examined. Upon impact, the fluid deforms and spreads out on the solid surface, leading to the formation of a thin film. This film then drags along with it a layer of the solid substrate. The motion of the fluid and the deformation of the solid substrate are complex and involve a variety of physical effects, including viscous flow, elastic deformation, and surface tension.

5:20PM AY.00005 Shock-wave solutions and their stability in two-layer channel flow1, ALIKI MAVROMOUSTAKI, RICHARD CRASTER, Imperial College London, OMAR MATAR — We study the dynamics of an interface separating two immiscible layers in an inclined channel. Lubrication theory is used to derive an evolution equation for the interface position that models the two-dimensional flow in both co- and counter-current configurations. This equation is parameterized by viscosity and density ratios, and a total dimensionless flow rate; the system is further characterized by the height of the interface at the channel inlet and outlet, which are treated as additional parameters. For one-dimensional flows, we use an entropy-flux analysis to delineate the existence of various types of shock-like solutions, which include compressive Lax-shocks, pairs of Lax and undercompressive shocks, and rarefaction waves. Criteria for the generation of shock-like solutions is employed to examine this fact, and to gain analytical insight about relevant aspects related to the stability of such exact stationary solutions.

1 EPSRC-funded studentship

9:05AM AY.00006 Numerical study of the evaporation of sessile drops: formation of hydrothermal waves1, KHELLIL SEFIANE, University of Edinburgh, GEORGE KARAPETNAS, Imperial College London, PEDRO SAENZ, PRASHANT VALLURI, University of Edinburgh, OMAR MATAR, Imperial College London — We study the family of steady shapes which arise when a magnetic liquid droplet is confined in a rotating Hele-Shaw cell, and subjected to an azimuthal magnetic field. Two different scenarios are considered: first, the magnetic field is assumed to be a Newtonian ferrofluid, and then it is taken as a viscoelastic magnetorheological fluid. The influence of the distinct material properties of the fluids on the ultimate morphology of the emerging stationary patterns is investigated by using a vortex-sheet formalism. Some of these exact steady structures are similar to the advanced time patterns obtained by existing time-evolving numerical simulations of the problem. A weakly nonlinear approach is employed to examine this fact, and to gain analytical insight about relevant aspects related to the stability of such exact stationary solutions.
9:18AM AY.00007 Thermal fluctuations and the breakup length of Savart capillary jets. F. JAVIER GARCÍA, Universidad de Sevilla, Spain — How long can a capillary jet be before it breaks up into droplets? Much time after the pioneering experimental work of Savart on the breakup of liquid jets isolated from external acoustic noise, no analytical prediction for their length has been derived yet. Even the precise nature of the perturbations leading to the natural breakup of a capillary jet remains a mystery. Only empirically fitted estimates have been proposed up to now, assuming an exponential growth of an unknown initial amplitude of those perturbations. Here, the evolution of a liquid jet emerging from a thin-wall orifice and subjected to thermal-noise fluctuations is explored through a stochastic linear modal analysis. Contrary to what has been assumed before, it is proven that the average amplitude of noisy perturbations does not grow exponentially. For the first time, a simple analytical estimate of the natural breakup length of a liquid jet is derived without the aid of any adjusting parameter. The breakup length of Savart liquid jets exiting through 3mm-diameter orifices are well predicted by this formula. The parametric range of application of this analysis and its accuracy are discussed.

1Supported by Ministerio de Ciencia y Tecnología under contract FIS2006-03645 and by Junta de Andalucía under contract FQM-421.


9:31AM AY.00008 Drift and symmetry breaking in Faraday waves. NICOLAS PERINET, PMMH-ESPCI-CNRS-LIMSI-UPMC, DAMIR JURIC, LIMSI-CNRS, LAURETTE TUCKERMAN, PMMH-ESPCI-CNRS, EDGAR KNOBLOCH, UC Berkeley — Faraday waves which break reflection symmetry and manifest horizontal flux have been investigated experimentally and theoretically by Fauve, Douady, Thual and by Knobloch, Martel, Vega. We perform the first numerical calculations of such states by means of fully-resolved simulations of the Navier-Stokes equations in superposed layers of air and water, coupled via a front-tracking algorithm for the interface. We find that these states are bistable with flux-free reflection-symmetric Faraday waves.

9:44AM AY.00009 Numerical and analytical studies of the electric field effects on interfacial waves subject to Rayleigh-Taylor instability. LYUDMILA BARANNYK, University of Idaho, DEMETRIOS PAPAGEORGIOU, Imperial College London, PETER PETROPOULOS, New Jersey Institute of Technology — A system of two stratified immiscible incompressible fluids in a horizontal channel of infinite extent is considered. Of particular interest is the case with the heavier fluid initially lying above the lighter fluid, so that the system is susceptible to the classical Rayleigh-Taylor instability. An electric field acting in the horizontal direction is imposed on the system and it is shown that it can act to completely suppress Rayleigh-Taylor instabilities and produces a dispersive regularization in the model. Dispersion relations are derived and a class of nonlinear traveling waves (periodic and solitary) is computed. Numerical solutions of the initial value problem of the system of model evolution equations that demonstrate a stabilization of Rayleigh-Taylor instability due to the electric field are presented. For weak electric fields, it is found that interface develops a finite-time singularity in the form of touchdown with the wall.

2The work of Lyudmyla Barannyk was partly supported by the University Research Council Seed Grant, University of Idaho. The work of Demetrios Papageorgiou and Peter Petropoulos was partly supported by the National Science Foundation grant DMS-0072228.

9:57AM AY.00010 Capillary wave motion excited by high frequency surface acoustic waves. MING TAN, Monash University, OMAR MATAR, Imperial College London, JAMES FRIEND, LESLIE YEO, Monash University — We present the results of a numerical and experimental study of capillary wave motion excited by high frequency surface acoustic waves (SAWs). A two-dimensional numerical model is constructed that couples the motion of the piezoelectric substrate to a thin liquid layer atop the substrate. A perturbation method, in the limit of small-amplitude acoustic waves, is used to decompose the equations governing fluid motion to resolve the widely differing time scales associated with the high frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra. The free frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra. The free frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra. The free frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra. The free frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra. The free frequency excitation. Transformation of time series data from both experiments and simulations into the frequency domain reveals that, in the low-amplitude regime, a fundamental resonant frequency, identical to that of the applied SAW, and a superharmonic frequency are found in the frequency spectra.

Sunday, November 21, 2010 8:00AM - 10:10AM – Session AZ Waves I Hyatt Regency Long Beach Regency F

8:00AM AZ.00001 Numerical and Experimental Study of Scott Russell’s Solitary Waves. JEONGWHAN CHOI, Korea University, SHU-MING SUN, Virginia Polytech, SANGHO OH, DALSOO LEE, KORDI, SUNG-IM WHANG, Ajou University, KOREA UNIVERSITY TEAM, VIRGINIA POLYTECH TEAM, KORDI TEAM — The motion of solitary waves on the free surface of a layer of water is studied. The waves are generated by a moving bump placed at the bottom or a pressure source on the surface. The problem is first discussed using a model equation, called force Kortweg-de Vries (FKdV) equation. Then, such forced waves are studied experimentally using a water tank with a moving bump at the bottom. The results from the FKdV equation match very well with those from the experiments if the solitary wave is not near the wave of maximum amplitude. Finally, it is shown that the solitary wave observed by Scott Russell in 1834 is just one of the forced solitary waves presented here.

8:13AM AZ.00002 Observation of odd and even two-dimensional standing solitary waves in water. JEAN RAJCHENBACH, ALPHONSE LEROUX, Universite de Nice- CNRS UMR 6622 — By means of the parametric excitation of water waves in a Hele-Shaw cell, we report the existence of two new types of highly localized, standing solitary water waves. They are respectively of odd and even symmetries. Both patterns oscillate subharmonically with the forcing frequency. The even pattern resembles the oscillon originally recognized at the surface of a vertically vibrated layer of brass beads [1]. The odd pattern has apparently never been observed before in any media.


8:26AM AZ.00003 Singularity formation in a model of shallow water wave equations. PRABIR DARIPA, Texas A&M University — We will present numerical solutions from initial value calculations of a model of shallow water wave equation. For small data, numerical solution develops singularity in a finite time. Driven by the structure of solutions, we carry out analysis based on numerical results to prove singularity formation. Numerical and theoretical results will be shown.
8:39AM AZ.00004 Generation of the maximum breaking wave amplitude by means of unidirectional wave focusing. IVAN SAVELYEV, ROBERT HANDLER, US Naval Research Laboratory — This work overviews existing methods and proposes new approach for the wave breaking generation in wave tanks. Due to dispersive nature of surface gravity waves, a unidirectional wave packet can be pre-programmed to focus its energy at a desired location in space and time. In this work, frequency modulated packets were generated by means of a single flat paddle hinged at the bottom. Two methods proposed within our approach reveal strong nonlinear wave envelope modulation in high amplitude regimes. Wave packets with high steepness were found to deviate from linear expectations by shifting their energy towards the leading edge. If left uncorrected, such modulation leads to defocusing of the wave energy, causing breaking waves to be less energetic and to appear prior to the desired location. Various empirical corrections were tested to account for the modulation, among which only the phase shift correction for the second method proved to be successful. Based on this finding, a wave generation procedure is established, which allows to simulate large wave breaking events and their interaction with various structures and vessels.

8:52AM AZ.00005 Experimental study of interfacial waves induced by surface waves in muddy water1, ERIC MAXEINIER, ROBERT A. DALRYMPLE, Johns Hopkins University — A peculiar feature has been observed in a laboratory tank with monochromatic surface waves propagating in muddy water with a thin layer of clear surface water: a quasi-stable set of interfacial waves that appear as longitudinally-oriented rotating tubes at the mud-water interface. These “interfacial tubes” are spatially periodic and temporally subharmonic structures whose direction of apparent rotation alternates with each passing surface wave crest. Rotation results from coupled upwelling and downwelling of clear surface water and muddy water below. The interfacial tubes appear to be standing nonlinear interfacial waves that result from a three-wave interaction involving a surface wave train and two interfacial wave trains. This is believed to be the first documented observation of this phenomenon in its nonlinear form. The topics covered in this presentation are relevant to the study of internal wave generation, wave damping and nearshore mixing processes.

The authors gratefully acknowledge support from the Office of Naval Research, Coastal Geosciences Program, and the Multidisciplinary University Research Initiative Grant: Mechanisms of Fluid-Mud Interactions Under Waves.

9:05AM AZ.00006 Near-critical reflection of nonlinear obliquely incident internal wave beam from a slope1, TRIANTAPHYLLOS AKYLAS, MIT — The reflection of internal gravity waves from sloping boundaries is believed to contribute significantly to vertical mixing in the ocean. This mechanism is likely to be enhanced when a wave is incident at an angle to the horizontal that is close to the slope of the boundary, given that the amplitude of the reflected wave becomes infinite according to linear inviscid theory if the angle of incidence exactly matches the slope. To clarify the role of nonlinear effects in this resonance, the reflection of a nonlinear wave beam of finite cross-section is analyzed by a matched-asymptotics approach, exploiting the fact that, near the critical angle, the reflected disturbance is confined to a thin boundary layer in an “inner” region close to the slope. Unlike prior studies, which assume that incident waves approach the boundary in a plane normal to the isobaths, here the oncoming wave is oblique. This gives rise to an alongslope mean flow component that is equally strong to the upslope induced mean flow, and the evolution of the reflected wave is fully nonlinear, in sharp contrast to the case of normal incidence where nonlinear effects are minor. The theoretical predictions are discussed in connection with related numerical and experimental results.

9:18AM AZ.00007 Laboratory experiments on internal wave reflection and absorption at a simulated oceanic pycnocline1, SCOTT WUNSCH, ALAN BRANDT, The Johns Hopkins University — Laboratory experiments have been performed to investigate the reflection of an internal wave beam with a “pycnocline” layer situated below an unstratified layer in order to simulate observed oceanic processes. An oscillating cylinder was used to generate wave beams in the well-known “St. Andrew’s Cross” pattern that interacted with the pycnocline. The internal waves were observed and the incident and reflected amplitudes measured using the synthetic schlieren technique. In virtually all instances, near-perfect reflection or near-complete absorption at the pycnocline was observed, depending on the value of the pycnocline density gradient. The data indicate the existence of a transition from reflection to absorption that is a function of the ratio of the maximum BV frequency in the pycnocline to the BV frequency of the stratified layer.

The authors gratefully acknowledge support from NSF.

9:31AM AZ.00008 The Near-Field Internal Wave Field Generated by a Sphere Moving in a Stratified Fluid1, JAMES ROTTMAN, KYLE BRUCKER, DOUGLAS DOMMERMUTH, SAIC, DAVE BROUTMAN, CPI — High resolution numerical simulations of a sphere traveling horizontally at constant speed at high Reynolds number through a uniformly stratified fluid are shown to compare well with previous laboratory experimental measurements of the drag and the internal wave field. The results of these detailed numerical studies are used to test and revise source distribution parameterizations of the near-field waves that have been used in analytical studies based on linear theory. Such parameterizations have been shown to be useful in initializing ray-tracing schemes that can be used efficiently to compute wave propagation through realistic oceans with variable background properties.

The authors gratefully acknowledge support from the Office of Naval Research (N00014-08-1-0290), Dr. Ron Joslin.

9:44AM AZ.00009 Horizontal transport of Lagrangian particles by basin-scale internal waves in a continuously-stratified circular lake, TAKAHIRO SAKAI, LARRY REDEKOPP, University of Southern California — Horizontal transport of Lagrangian particles under the influence of wind-generated, basin-scale internal waves in a circular lake is studied by employing field solutions of the linear hydrostatic model and by simulations of the weakly-nonlinear, weakly-nonhydrostatic evolution model for continuous stratification subject to wind stress forcing over a basin of uniform depth. Both the azimuthal mode-one Kelvin and the gravest Poincare waves of the first two vertical eigenmodes are accounted for in the specification of the advection field. It is found that Kelvin waves play the dominant role in along-shore transport. Although vertical mode-two (V2) waves are usually considerably smaller than those of vertical mode-one (V1), yet V2 Kelvin waves possess sufficient ability in driving along-shore transport for a distance comparable with that of V1-only transport. This arises because of the longer residence time of disparately slow V2 Kelvin waves confined in the vicinity of the basin perimeter. Poincare waves, on the other hand, play a dominant role in off-shore transport, stretching and squeezing the particle cloud in radial directions with fast, near-inertial frequencies. These transport features are compared for different wind forcing strengths and different horizontal scale of the basin.

1Supported by the Office of Naval Research (contract number N00014-08-C-0508).

1Supported by NSF.
focus on the evolution of dissipation rate (Turbulent mixing of a passive scalar (temperature) is studied by means of experiments and numerical simulations in turbulent channel flow, with an emphasis on the scalar dissipation rate ($\varepsilon$)). The scalar (temperature) is injected at small scales by a heated line source, aligned in the spanwise direction. The present experiments focus on the evolution of $\varepsilon$ downstream of the line source, for different wall-normal source locations. In particular, knowledge of the different components of $\varepsilon$ ($\partial^2 \theta / \partial x^2$, $\alpha(\partial \theta / \partial y)^2$, and $\alpha(\partial^2 \theta / \partial z^2)$), where $\alpha$ is the thermal diffusivity) enable the quantification of the small-scale passive scalar statistics, and their (presumed) return to isotropy from an initially anisotropic injection. Measurements of temperature derivatives were performed by means of cold-wire thermometry. A direct numerical simulation was also undertaken to provide complementary data, difficult to obtain experimentally. The velocity field was independently computed using the freely-available channel flow code of Dr. John Gibson (http://www.channelflow.org). The advection-diffusion equation was solved using a third-order scheme with the flux integral method (Leonard, Appl. Math. Modelling, 1995).

ARPI BERAJEKLIAN, LAURENT MYDLARSKI, McGill University — The principal objective of this work is to study the sensitivity of (i) the turbulent Prandtl number ($P_{TR}$), and (ii) the mechanical-to-thermal time-scale ratio ($r$) to differences in the scalar field's injection method within the same (hydrodynamic) flow. Both are recurring quantities employed in turbulence models, determined from experiments, and generally assumed to be (flow-dependent) constants. To this end, mixed velocity-temperature measurements were made in the heated wake of a circular cylinder. The passive scalar under consideration was temperature and the wake was heated by one of two ways: heating the cylinder itself, or by use of a mandrel placed downstream of the cylinder. For each case, the distributions of the turbulent Prandtl number and the mechanical-to-thermal time-scale ratio were compared. The experimental results demonstrate that both $P_{TR}$ and $r$ differ for the two scalar injection methods (in addition to varying across the wake). Hence, both $P_{TR}$ and $r$ not only depend on the type of flow, but on the scalar field injection method as well - a result that is generally not taken into account when turbulent flows are modeled.

This work is supported by the Metström Program of the DFG.

1Support has been graciously provided by the NSERC (Canada).

1:30PM CA.00004 Experiments and simulations of passive scalars released from concentrated sources in turbulent channel flow1, EMMANUEL GERMAINE, LUCA CORTELEZZI, LAURENT MYDLARSKI, McGill University — Turbulent mixing of a passive scalar ($\theta$) is studied by means of experiments and numerical simulations in turbulent channel flow, with an emphasis on the scalar dissipation rate ($\varepsilon$)). The scalar (temperature) is injected at small scales by a heated line source, aligned in the spanwise direction. The present experiments focus on the evolution of $\varepsilon$ downstream of the line source, for different wall-normal source locations. In particular, knowledge of the different components of $\varepsilon$ ($\partial^2 \theta / \partial x^2$, $\alpha(\partial \theta / \partial y)^2$, and $\alpha(\partial^2 \theta / \partial z^2)$), where $\alpha$ is the thermal diffusivity) enable the quantification of the small-scale passive scalar statistics, and their (presumed) return to isotropy from an initially anisotropic injection. Measurements of temperature derivatives were performed by means of cold-wire thermometry. A direct numerical simulation was also undertaken to provide complementary data, difficult to obtain experimentally. The velocity field was independently computed using the freely-available channel flow code of Dr. John Gibson (http://www.channelflow.org). The advection-diffusion equation was solved using a third-order scheme with the flux integral method (Leonard et al., Appl. Math. Modelling, 1995).

1Support has been graciously provided by the NSERC (Canada).

1:52PM CA.00005 Phase relaxation of a cloud water droplet ensemble undergoing turbulent mixing1, BIPIN KUMAR, TU Ilmenau, Germany, RAYMOND A. SHAW, Michigan Tech, USA, JOERG SCHUMACHER, TU Ilmenau, Germany — The understanding of the entrainment and mixing of clear (sub saturated) with cloudy air at the boundary of a cloud is still far from being complete. Mixing is determined by the ratio of two time scales: the mixing time and the phase relaxation time, which can be combined as a Damköhler number. The phase relaxation time is connected with the water phase change and thus changes in the cloud water droplet size distribution and their number density. The mixing time of the advecting turbulent flow is determined by the size and velocity of the turbulent eddies. Here, we will outline a direct numerical simulation model that couples the Eulerian description of the velocity and water vapor fields with a Lagrangian ensemble of cloud water droplets. The simulations resolve a small cubic fraction of the cloud and simulate a homogeneous isotropic turbulent flow. Turbulence properties at larger scales are taken from field measurements of the helicopter-based measurement platform ACTOS. Cloud water droplets can grow and shrink, as determined by the advected vapor concentration field that sets the local supersaturation at the droplet position. First results of our direct numerical simulations are presented.

1This work is supported by the Metström Program of the DFG.
2:05PM CA.00006 Direct Computation of Two- and Three-dimensional FTLE/LCS from Particle Tracking Velocimetry Data, SAMUEL RABEN, ROD LA FOY, SHANE ROSS, PAVLOS VLACHOS, Virginia Tech — Finite-Time Lyapunov Exponents (FTLEs) and Lagrangian Coherent Structures (LCSs) are becoming more commonly utilized for the interpretation of unsteady experimental flow fields. FTLEs provide information on regions of high attraction, repulsion, and shear in a flow field and can be used to investigate transport and mixing. Elevated values in FTLE fields, or ridges, can be evaluated in time and are what are referred to as LCS. In order to compute the FTLE field from velocity fields, typically artificial particles are seeded into the field and then numerically integrated to find positions in time from the given velocity information. This process can be very computationally expensive. When dealing with experimental data such as PIV or PTV it is possible to decrease the computational cost by simply tracking the flow tracers already present in the flow, avoiding the additional steps of inferring the velocity and artificial particle seeding. Through the use of Lagrangian particle tracking this work finds that it is more computationally efficient, as well as more accurate, to calculate FTLEs this way. This work considers both 2D as well as 3D flow fields for this analysis.

2:18PM CA.00007 Flow around finite-size neutrally buoyant Lagrangian particles in fully developed turbulence1, MATHIEU GIBERT, SIMON KLEIN, ANTOINE BÉRUT, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — By using an innovative technique based on Lagrangian Particle Tracking (LPT), we have been able to follow the motion of finite-size neutrally buoyant particles together with the trajectories of tracer particles in the surrounding fluid. The particles we study have diameters of about 200 times the dissipative scale of the flow, and their density is almost that of the fluid. The experiments are conducted in a von Karman swirling water flow at Taylor microscale Reynolds numbers up to 500. By measuring the full motion of the big particles (translation and rotation), we are able to “sit” in their frame of reference and measure the flow properties around them. We will report experimental results on the flow properties and its correlations with the big particle trajectories in this Lagrangian frame.

1This work was funded by the Max Planck Society, and the Marie Curie Fellowship, Programme PEOPLE - Call FP7-PEOPLE-IEF-2008 Proposal No. 237521.

2:31PM CA.00008 Measurements of 3D velocity and scalar field for a film-cooled airfoil trailing edge1, MICHAEL BENSON, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Turbine blade tips commonly are cooled by venting air through slots upstream of the trailing edge. The effectiveness of this approach is governed by the rate of mixing of the coolant with the mainstream flow, which is strongly under-predicted by conventional RANS models. Experiments were conducted for a simple airfoil with a modern trailing edge cooling geometry. The full 3D coolant concentration distribution was measured using Magnetic Resonance Imaging (MRI). The scans measured the concentration distribution with a spatial resolution of 0.5 mm3 and an uncertainty near 5%. Magnetic Resonance Velocimetry (MRV) was used to provide 3D, mean velocity measurements in the identical flow. Blowing ratios of 1.0, 1.3, and 1.5 were examined at Reynolds numbers of 50,000 and 100,000 based on airfoil chord length. The coupled concentration and velocity measurements were used to develop a qualitative picture of the flow structures contributing to the rapid mixing. Surface concentration measurements provide film cooling effectiveness data, which were compared for validation purposes with traditional thermal measurements. The MRI-based technique for measuring film cooling effectiveness avoids the large uncertainties caused by conduction in the thermal tests.

1Work sponsored by General Electric and the Army Research Office.

2:44PM CA.00009 Coupled Velocity and Cooling Effectiveness Measurements of a Film Cooling Hole With Varied Blowing Rates and Ejection Angles, EMIN ISSAKHANIAN, CHRIS J. ELKINS, JOHN K. EATON, Stanford University — Film cooling is used to shield turbine blades from combustion gases which are at temperatures above the melting point of the blade’s constituent alloy. Maximizing film cooling effectiveness allows higher combustion temperatures and decreases need for bypass air. The present experiment studies flow through a single film cooling hole jetting into a square channel. The momentum thickness Reynolds number of the main flow is 500. The diameter of the cooling flow is 10 times the momentum thickness at the hole exit. The cooling flow Reynolds number varies between 1250 and 5000. Magnetic Resonance Velocimetry (MRV) and Concentration (MRC) are used to measure mean velocity and coolant concentration of the 3-D field both inside the main channel and inside the cooling hole and feed plenum. By marking only the main flow with a passive scalar, the MRC data allow measurement of cooling flow concentration, which by analogy is related to the temperature of the fluid. The velocity data shows the development of a counter-rotating vortex pair downstream of the jet. These vortices transport cooling flow away from the channel floor resulting in a lifted kidney-shaped coolant cross-section and reduced effectiveness. The varying strength of this flow feature and of surface effectiveness due to different ejection angles and blowing ratios is studied.

2:57PM CA.00010 The highly excited confined mixing layer, WEI ZHAO, GUIREN WANG, University of South Carolina, Columbia — In order to understand the mechanism of ultrafast mixing observed in confined mixing layer in a pipe, flow velocity and vorticity fields are quantitatively measured with PIV. Under strong forcing, the responses of the velocity and vorticity field are completely different, leading to a different mixing process. In free mixing layer, under strong forcing level, there will be saturation, and the initial mixing depends only on the convective scalar transport by velocity fluctuations of large scale spanwise vortex. However in confined mixing layer under strong forcing, it is found that not only the velocity fluctuations play important roles in mixing process, but also the mean velocity field. The local averaged vertical velocity V and spanwise velocity W are not negligible anymore and can be significantly affected by the forcing. These mean velocity constituents have two direct effects: One is to transport scalar outward from the axial line, which leads to a larger spreading rate. Another is to accelerate the breakdown of vortex structures by stretching. Hence, the mixing can be significantly enhanced. From the distributions of velocity and vorticity, it can be found that both the corner vortex and strongly asymmetric influence of actuating coexist. When the forcing amplitude is increased to 11%, the instant vortex structures near the pipe axis indicate a high consistency with the mean vortex structures. Hence, at high forcing level, we tend to believe that the corner vortex is the primary source of V constituent.

Sunday, November 21, 2010 1:00PM - 3:10PM
Session CB Turbulent Boundary Layers II  Long Beach Convention Center 101B

1:00PM CB.00001 Volumetric Velocimetry in Wall Bounded Flows, TRISTAN CAMBONIE, JEAN-LUC AIDER, Laboratoire PMMH - UMR7636 CNRS - ESPCI - University Paris 6 - University Paris 7, “INSTABILITY, CONTROL AND TURBULENCE” (ICT) TEAM — The role of three-dimensional (3D) perturbations in the transition process of boundary layers is well-known. One of the major limitations in the understanding of the complex mechanisms involved in the transitions relies on the experimental difficulty in investigating strongly 3D unstationary flows. In this study, we apply for the first time Volumetric Velocimetry to a boundary layer flow. Volumetric Velocimetry gives access to the instantaneous three-components (3C) 3D velocity field (3D3C). It has already been applied to various detached vortical structures like vortex rings, but it is still a challenge in wall bounded flows because of the need of a good spatial resolution to resolve small-scale structures and because of the optical perturbations induced by the wall. We propose a dedicated experimental methodology adapted to these constraints and apply it successfully to complex 3D flows induced by different three-dimensional perturbations introduced into a flat-plate boundary layer.
1:13PM CB.00002 Corrected Hot-wire Measurements of Stream-wise Turbulence Intensity from Several ZPG Boundary Layers¹, HASSAN M. NAGIB, RICHARD D. DUNCAN, IIT, Chicago, PETER A. MONKEWITZ, EPFL, Switzerland — Current experimental activity aimed at resolving the scaling of stream-wise turbulence intensity profiles $\overline{\tau(y)}$ with Reynolds number in turbulent flat plate boundary layers has brought the largely unresolved issue of correcting systematic errors in hot-wire measurements of $\overline{\tau(y)}$ into focus. Recently, we demonstrated the effectiveness of a heuristic scheme to generate unique $\overline{\tau(y)}$ profiles from selected data sets obtained with single hot-wires of widely different length, aspect ratio and construction over a large Reynolds number range of $4,000 < Re_{\delta} < 50,000$. The scheme has been applied to a larger number of data sets in zero pressure gradient (ZPG) boundary layers and is used here to re-examine the scaling of the stream-wise turbulence intensity profiles. These results confirm or reveal Reynolds number trends of features such as the peak intensity, in addition to pinpointing limitations of some of the data sets in the literature. The best scaling for $\overline{\tau(y)}$ is examined in different parts of the boundary layer as a function of Reynolds number.

¹Partially Supported by the Illinois NASA Space Grant Consortium.

1:26PM CB.00003 Resolving the 3D velocity field inside a Roughness Sublayer in a turbulent channel flow using HPIV¹, SIDDHARTH TALAPATRA, JOSEPH KATZ, Johns Hopkins University — Microscopic holographic PIV is used to measure the 3D velocity field within the roughness sublayer of a turbulent channel flow at $Re_{\delta}$ of 3400. Recording holograms through a rough surface is facilitated by matching the optical refractive index of the rough wall with that of the working fluid, a concentrated solution of NaI in water. The pyramidal roughness height is $k=0.45$mm, the sample volume size is $3.2 \times 1.8 \times 1.8$mm³, the long dimension being in the streamwise direction, and the wall-normal range is $-0.33 < y/k < 3.67$, where $y=0$ is located at the roughness peak. The flow is locally seeded with 2μm particles, and in the current data, 5000 particles are tracked per hologram pair. The resulting unstructured vectors are interpolated onto a regular 3D grid to obtain vectors with a spacing of 60μm or 8.5 wall units. The data show that $\alpha y/k < 0.5$, there is a preferred channeling of the flow along paths that circumvent the pyramid crest lines. Planar vorticity distribution from different perspectives as well as 3D isosurfaces show that the near wall region is flooded by quasi-streamwise vortices that are aligned at shallow angles and have a typical streamwise extent of 1-2k.

¹Supported by ONR (grant No. 000140-91-0-07-7)

1:39PM CB.00004 A reinterpretation of the distribution of vortical structure in wall turbulence¹, BEVERLEY MCKEON, IAN JACOBI, California Institute of Technology, ATI SHARMA, Imperial College London — The critical layer framework for turbulent pipe flow proposed by McKeon & Sharma (J. Fluid Mech. 2010, see also the DFD-2010 presentation on ‘Structure from the critical layer framework in turbulent flow’ by Sharma & McKeon) provides a model by which the distribution of hairpin-like vortices in wall turbulence can be reinterpreted. The model is used to demonstrate that the shear associated with the wall-normal variation of the mean velocity profile suppresses so-called retrograde vortices. Analysis of PIV images in the streamwise/wall-normal plane of a relatively low Reynolds number, zero pressure gradient turbulent boundary layer shows that the use of a Galilean invariant decomposition of the velocity field prior to the calculation of swirl, i.e. subtraction of a constant convection velocity from the field, leads to a distorted count of the apparent number of retrograde spanwise vortices. By comparison, the result of a Reynolds decomposition suggests an essentially even distribution between prograde vortices (identified by many researchers as the heads of hairpin vortices) and retrograde ones, (i.e. vortices with the opposite sense of rotation) in agreement with the critical layer model.

¹This research is sponsored by the AFSOR (program manager J. Schmisser) and an Imperial College Junior Research Fellowship.

1:52PM CB.00005 Non-intrusive measurements in hypersonic shock wave turbulent boundary layer interactions¹, ANNE-MARIE SCHREYER, Institute for Aerodynamics and Gas Dynamics, University of Stuttgart, 70569 Stuttgart, Germany, DIFÉNKR AHOO, ALEXANDER J. SMITS, Mechanical and Aerospace Engineering, Princeton University, Princeton, NJ 08544 — Measurements are presented to describe the behavior of a shock wave turbulent boundary layer at hypersonic speed. Mean flow and two components of velocity fluctuations are obtained using PIV, and flow visualization and thermal imaging are performed using schlieren and infrared imaging, respectively. The boundary layer develops at Mach 7.2 in a perfect gas, at a Reynolds number based on momentum thickness between 3600 and 5500. The shock waves are created using two-dimensional compression corners of 8° and 33°. The flow field is attached for the 8° corner, and separated for the 33° corner. The modification of the flow field due to micro-ramps placed upstream of the 33° corner is also discussed.

¹Supported under NASA Cooperative Agreement NNX08AB46A.

2:05PM CB.00006 Fully-resolved turbulence measurements in high Reynolds number pipe flow using a nano-scale probe¹, MARGIT VALLIKIIVI, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — Statistics of the streamwise velocity component were measured at Reynolds numbers from $45 \times 10^3$ to $800 \times 10^3$ in the Princeton/ONR Superpipe. A nano-scale thermal anemometry probe (NSTAP) with sensing volume $60 \times 2 \times 0.1\mu m$ was used to obtain data that were free from the effects of spatial filtering. The results agree well with data from conventional measurement techniques for the lower Reynolds numbers but the NSTAP allows the study of fully resolved turbulence at Reynolds numbers almost one order of magnitude larger than conventional techniques. The data gives new insight on pipe flow turbulence, as well as providing a quantitative estimate of the effects of spatial filtering, allowing for re-evaluation of previous datasets obtained with limited spatial resolution.

¹Supported under NSF Grant CTS-0421147 and ONR Grant N00014-09-1-0263.

2:18PM CB.00007 Non-Local Geometry of High Reynolds Number Wall-Bounded Turbulence using Tomographic Particle Image Velocimetry. CALLUM ATKINSON, Laboratory Turbulence Research in Aerospace and Combustion, MICHEL STANISLAS, Laboratoire de Mecanique de Lille, JULIO SORIA, Laboratory Turbulence Research in Aerospace and Combustion — Three-dimensional fields resulting from tomographic particle image velocimetry (Tomo-PIV) measurements in the buffer region of a fully developed flat plate turbulent boundary layer at $Re_\delta = 7800$ and 11800 are examined to extract statistics on the geometry and spatial distribution of large-scale coherent structures in wall bounded turbulence. Fields of $470^\perp \times 70^\perp \times 470^\perp$ and $920^\perp \times 140^\perp \times 920^\perp$ wall units are examined at a distance of $y^+ = 8$ to 55 and $y^+ = 15$ to 100 from the wall for $Re_\delta = 7800$ and 11800, respectively. Pattern recognition methods are used to extract and classify the structures. Results are compared with Stereo-PIV and direct numerical simulations of a boundary layer at $Re_\delta = 1900$. The enstrophy, dissipation and kinetic energy associated with each of these structures are considered.
the Rapid Distortion Theory of Turbulence modeling for all hybrid methods. The modeling of the related turbulent transport term is also considered. Although this work in performed in the PANS context, its findings can benefit closure of the unresolved stresses (second-order central moments) and dissipation, as well as the resolved fields subject to different cut-off lengthscales are examined.

method. In the PANS method, the cut-off resolution is parameterized in terms of ratios of unresolved-to-resolved kinetic energy and dissipation. The behavior resolution turbulence modeling approaches. In this work, we revisit the log-law analysis in the context of Partially Averaged Navier-Stokes (PANS) bridging, DASIA REYES, SHARATH GIRIMAJI, Texas A&M University — High-fidelity near-wall modeling is crucial for the success of all variable-resolution turbulence simulation schemes. The predictions of the new model are evaluated in a variety of canonical test cases.

The downstream flow field is reconstructed in a phase-locked sense in order to compare the observed behavior with asymptotic representations of the expected behavior at matched flow conditions. Perturbation using a periodic disturbance is shown to reveal underlying features of the turbulent boundary layer which are intimately connected to the critical layer framework for turbulent pipe flow proposed by McKeon & Sharma (see the DFD-2010 presentation on ‘Structure from the critical layer framework in turbulent flow’ by Sharma & McKeon), while simultaneously providing practical insight on the manipulation of the structure of boundary layers.

1:26PM CC.00003 Bridging Approach to variable-resolution turbulence simulations – Unification of PANS and PITM closures , SHARATH GIRIMAJI, Texas A&M University, ROBERT RUBINSTEIN, NASA Langley Research Center — Variable-resolution turbulence simulation schemes can be classified into two general categories: hybrid methods and bridging approaches. Hybrid computations entail Reynolds averaged Navier-Stokes (RANS) calculations in some flow regions and large eddy simulations (LES) in others. There exists no clear consensus on the criterion for switching from RANS to LES. The bridging methods, on the other hand, seamlessly transition from one flow resolution to another. There are currently two major bridging approaches – Partially-Averaged Navier Stokes (PANS) and Partially Integrated Turbulence Model (PITM). While the two bridging closure expressions share many similarities, there are notable differences as well. In this work, we generalize the PITM derivation to include additional features and demonstrate complete consistency with PANS closure. The unification of the two methods confirms the mathematical rigor of the bridging approach and highlights its advantage over hybrid methods.

1:39PM CC.00004 Modeling near-wall turbulence behavior for variable-resolution bridging methods , DASIA REYES, SHARATH GIRIMAJI, Texas A&M University — High-fidelity near-wall modeling is crucial for the success of all variable-resolution turbulence modeling approaches. In this work, we revisit the log-law analysis in the context of Partially Averaged Navier-Stokes (PANS) bridging method. In the PANS method, the cut-off resolution is parameterized in terms of ratios of unresolved-to-resolved kinetic energy and dissipation. The behavior of the unresolved stresses (second-order central moments) and dissipation, as well as the resolved fields subject to different cut-off lengthscales are examined. The modeling of the related turbulent transport term is also considered. Although this work in performed in the PANS context, its findings can benefit closure modeling for all hybrid methods.

Sunday, November 21, 2010 1:00PM - 3:10PM –
Session CC Turbulence Modeling II Long Beach Convention Center 102A

1:00PM CC.00001 ABSTRACT WITHDRAWN –
Evaluation of Partially-Averaged Navier-Stokes (PANS) bridging method in turbulent channel flows, BRANISLAV BASARA, AVL GMBH, Graz, Austria, SHARATH GIRIMAJI, Texas A&M University, PAVLOVIC PAVLOVIC, AVL GMBH, Graz, Austria — The Partially-Averaged Navier-Stokes (PANS) variable-resolution turbulence computational method is intended for seamless bridging between Reynolds-Averaged Navier-Stokes (RANS) and Direct Numerical Solution (DNS). While the success of PANS has been well documented in separated flows with largescale instabilities, its performance in wall-bounded flows is yet to be demonstrated. Toward that end, we perform channel flow simulations using the PANS $\zeta$-$f$ closure which is based on the near-wall RANS $\zeta$-$f$ model. The filter width is controlled by specifying the appropriate control parameters, unresolved-to-total ratios of turbulent kinetic energy and frequency. Computations are performed with three different grids (filter-widths) for the case of $Re_\tau = 650$. Direct Numerical Simulation and Large eddy simulation data are used for comparison and evaluation. The results clearly demonstrate that PANS performs well in near-wall turbulent flows as well.

Turbulence Closures Uncertainties in Shock Boundary Layer Interactions, MICHAEL EMORY, RENE PECNIK, GIANLUCA IACCARINO, Stanford University — Reynolds-averaged closures have limited predictive capabilities when applied to the problem of shock boundary layer interaction. Several modifications to RANS models have been proposed in the literature, including compressibility corrections, limiters, and alternative forms of the turbulence production terms. Our objective is to characterize the errors introduced by the various approximations used in typical two-equation models, such as turbulence isotropy, linear stress-strain relationship, dissipation rate, etc. by isolating each contribution separately. We use the barycentric map to alter the turbulence anisotropy and introduce realizable Reynolds stress perturbations to investigate the effect of modeling errors on the resulting wall pressure and shock position. This method of measuring structural uncertainty is compared against two traditional uncertainty quantification approaches which evaluate the effect of boundary condition and model coefficient variability on the quantities of interest. We also discuss how appropriate input uncertainty ranges are determined.

Adaptive wall functions for moving walls using the $k-\omega$ turbulence model, JOHN AXERIO-CILIES, GIANLUCA IACCARINO, Stanford University — An adaptive wall function for the $k-\omega$ model is derived for moving walls starting from a wall-resolved RANS computation of the flow over a moving flat plate with zero pressure gradient. The wall function is implemented via lookup tables for the turbulence quantities and the friction velocity $u_r$. The reference well-resolved, grid-converged RANS numerical solutions are obtained using the $k-\omega$ turbulence model with wall integration on very fine grids ($y^+ < 1$). Selecting a reference frame such that $U_\infty \geq 0$ yields three distinct velocity profile regimes: $U_m \leq 0 < U_w$, $U_m \geq 0 > U_w$, and $U_m > U_w > U_\infty$. It is shown that adaptive wall functions are appropriate for all three velocity profile regimes as well as different Reynolds numbers when the near wall grid resolution is not sufficient ($y^+ > 1$). For very fine grids ($y^+ < 1$) this approach yields results consistent with the wall integration solution. Finally, the performance of the proposed adaptive wall functions is investigated for the complex flow around a rotating Formula 1 tire. The complexity of this flow arises from the impingement and jetting at the front of the tire, strong pressure gradients, and the large separated region behind the tire.

Two-Equation Turbulence Model Predictions of Transition in Buoyancy-and Shock-Driven Flows, BRYAN JOHNSON, OLEG SCHILLING, Lawrence Livermore National Laboratory — Two-equation Reynolds-averaged Navier-Stokes (RANS) models are generally regarded as relevant only for fully-developed turbulent flow. It is shown here that the early-time evolution of these models captures the turbulent transition for Rayleigh–Taylor instability and shock–turbulence interaction. When the fluctuation energy is much less than the mean flow energy, turbulent diffusion is negligible and the equations can be integrated analytically for a steady mean flow. For an incompressible flow, the turbulent kinetic energy grows exponentially at the physical growth rate (with appropriate model coefficients). For a shock-driven flow, the turbulent kinetic energy is amplified over the advection time across the shock, with an amplification factor equivalent to the physical amplification factor. Once turbulent diffusion becomes important, the turbulent quantities across the mixing layer are generally insensitive to the initial evolution. The primary consequence of varying model coefficients and initial conditions in the linear regime is a shift in the time at which the mixing layer begins to develop.

Bayesian Calibration and Comparison of RANS Turbulence Models for Channel Flow, TODD OLIVER, ROBERT MOSER, University of Texas — A set of RANS turbulence models—including Baldwin-Lomax, Spalart-Allmaras, $k-\epsilon$, and $\omega^2-\omega$—are calibrated and compared in the context of fully-developed channel flow. Specifically, a Bayesian calibration procedure is applied to infer the parameter values for each turbulence model from channel flow DNS data. In this process, uncertainty arises both from uncertainty in the data and inadequacies in the turbulence models. Various stochastic models of the turbulence model inadequacy are formulated, and the impacts of different uncertainty modeling choices are examined. The calibrated turbulence models are compared in terms of two items: posterior plausibility and predictions of quantities of interest such as centerline velocity and the location of the maximum Reynolds shear stress. The posterior plausibility indicates which model is preferred by the data according to Bayes' theorem, while the predictions allow assessment of how strongly the model differences impact the quantities of interest. The implications of these comparisons for turbulence model validation will be discussed. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

Reproducing second order statistics of turbulent flows using linearized Navier-Stokes equations with forcing, MIHAILO JOVANOVIC, TRYPHON GEORGIOU, University of Minnesota — We study the problem of reproducing second order statistics of turbulent flows using linearized Navier-Stokes (NS) equations with forcing. This forcing is represented by a stochastic excitation that enters into the equations as an additive spatio-temporal body force. For homogeneous isotropic turbulence, we show that the steady-state velocity correlation tensors can be exactly matched by the linearized NS equations subject to a temporally white solenoidal forcing with appropriately selected second order statistics. For turbulent channel flows, however, forcing of the linearized equations by colored-in-time stochastic process is required. The forcing spectra, which are consistent with DNS data, are obtained from a solution of the maximum entropy optimization problem. We show how this forcing can be generated as an output of a spatio-temporal filter driven by white-in-time stochastic process with appropriately selected second order statistics. Our results can be used to model forcing correlations in, (i) receptivity analysis of a free-stream turbulence induced transition in boundary layers and (ii) design of flow estimators and controllers for turbulence suppression in wall-bounded shear flows.

Part of this work was performed during the 2010 Summer Program at the Center for Turbulence Research at Stanford University.

Sunday, November 21, 2010 1:00PM - 2:57PM – Session CD CFD: Algorithms II Long Beach Convention Center 102B
1:00PM CD.00001 An Adaptive Mesh Refined Gradient-Augmented Level Set Method1, JEAN-CHRISTOPHE NAVE, McGill University, BENJAMIN SEIBOLD, Temple University, RUBEN ROSALES, MIT — The Gradient-Augmented Level Set method (GA-LS) was introduced at the 62\textsuperscript{nd} annual APS-DFD meeting by Nave et al. (arXiv:0905.3409). Leveraging the optimal locality and unconditional stability of the method, we present a natural extension to adaptive quad-tree meshes. The new method possesses many desirable features such as improved mass conservation, reduced computational effort, and is, due to the optimal locality property of the underlying GA-LS, very easy to implement. Several key benchmark tests will be presented to help demonstrate the benefits of the approach, and the overall simplicity of the algorithm.

1This research was supported by NSF grant DMS-0813648.

1:13PM CD.00002 Adaptive mesh refinement for large-eddy simulation using the dynamic reconstruction model. LAUREN GOODFRIEND, FOTINI CHOW, UC Berkeley, MARCOS VANELLA, ELIAS BALARAS, University of Maryland — Combining large-eddy simulation (LES) with adaptive mesh refinement (AMR) to reduce computation costs yields a powerful technique for modeling flows with complex geometries. Increased errors associated with the variable mesh sizes used in AMR have limited computations combining these methods. Using explicit filtering to separate turbulent stresses into resolvable sub-filter scale (RSFS) and sub-grid scale (SGS) terms may help control these errors. In this study, the dynamic reconstruction model (DRM) is used to approximate the RSFS stress using a series expansion to invert the explicit filter operation, with a dynamic eddy viscosity model for the SGS stress. DRM has previously been shown to reduce numerical truncation errors, leading to a more accurate turbulence closure. Here, the effect of using discontinuous versus continuously varying filter width across a grid refinement interface is investigated in the context of the DRM. Decaying isotropic turbulence is advected past a refinement interface in which the grid is coarsened. Results are compared using explicit filtering with different levels of reconstruction and without using explicit filtering.

1:26PM CD.00003 Characteristic Boundary Conditions with Transverse Effects: a Comparative Study. GUIDO LODATO, HEINZ PITSCH, CTR, Stanford University — The inclusion of transverse effects in designing characteristic boundary conditions for the Euler and compressible Navier-Stokes equations was discussed by many authors and proved to give significant improvement in reducing numerical perturbations from open boundaries. Starting from the characteristic formulation of the Euler equations using generalized coordinates, an analysis of the different transverse terms is carried out. Based on the identification of different types of transverse effects, it is suggested that, in order to achieve the best performance for the numerical behavior of the boundary conditions, different transverse terms should be treated differently. The analytical check on well-posedness and reflection coefficients gives evidence that stability might not be an issue regardless of the way these terms are treated, but that numerical reflection from the boundary might be negatively affected by the occurrence of a direct coupling between outgoing vorticity modes and incoming acoustic modes. This analysis is supported by numerical tests on the inviscid convected vortex problem at different Mach numbers.

1:39PM CD.00004 Multi-core/GPU accelerated multi-resolution simulations of compressible flows. BABAK HEJAZIALHOSSEINI, DIEGO ROSSINELLI, PETROS KOUMOUTSAKOS, Chair of Computational Science, ETH Zurich, CH-8092, Switzerland — We develop a multi-resolution solver for single and multi-phase compressible flow simulations by coupling average interpolating wavelets and local time stepping schemes with high order finite volume schemes. Wavelets allow for high compression rates and explicit control over the error in adaptive representation of the flow field, but their efficient parallel implementation is hindered by the use of traditional data parallel models. In this work we demonstrate that this methodology can be implemented so that it can benefit from the processing power of emerging hybrid multicore and multi-GPU architectures. This is achieved by exploiting task-based parallelism paradigm and the concept of wavelet blocks combined with OpenCL and Intel Threading Building Blocks. The solver is able to handle high resolution jumps and benefits from adaptive time integration using local time stepping schemes as implemented on heterogeneous multi-core/GPU architectures. We demonstrate the accuracy of our method and the performance of our solver on different architectures for 2D simulations of shock-bubble interaction and Richtmeyer-Meshkov instability.

1:52PM CD.00005 Stability and Accuracy of Coupling Strategies in Hybrid LES/PDF Algorithms for Turbulent Reactive Flows. PAVEL POPOV, HAIFENG WANG, STEPHEN POPE, Cornell University — Hybrid Large Eddy Simulation/Probability Turbulence Density Function (LES/PDF) algorithms for turbulent reactive flow consist of two main components: a grid-based LES solver, and a Monte Carlo solver which approximates the composition PDF via an ensemble of Lagrangian particles. As a part of the interplay between these two codes, the PDF solver needs to access the LES solver information which it needs in order to evaluate the filtered density field. Here, we assess two alternative strategies for implementing this coupling: either by directly passing density information from the PDF to the LES solver, or by evaluating density from an enthalpy field which is solved for by the LES code, and relaxed towards a consistent (up to numerical errors) PDF-based enthalpy field. We compare the stability of these two approaches, and their accuracy, defined as the level of consistency between a standalone LES and a coupled LES/PDF solution for a laboratory-scale jet flame, with flamelet chemistry modeling. The benefits of second-order time accuracy, relative to a first-order-accurate in time implementation, are examined, and different approaches for performing filtering of the PDF fields, which contain significant noise due to their stochastic nature, are evaluated.

2:05PM CD.00006 Theoretically Simple Thermal Lattice Boltzmann Method and Its Application to a 2-D Shock Tube Simulation. JAE WAN SHIM — We propose a new approach to derive the thermal lattice Boltzmann method (LBM), and show its accuracy and stability by a 2-D shock tube simulation. The derivation is as simple as that of the isothermal LBM obtained by using Taylor expansion. We do not use Gauss-type quadratures because the results are inapplicable in regular lattices for thermal flows. The derivation enables us to simulate thermal flows with accuracy, stability, and use of regular lattices, simultaneously.

2:18PM CD.00007 Shape Optimization for Drag Reduction in Linked Bodies using Evolution Strategies and the Hybrid Wavelet Collocation – Brinkman Penalization Method. OLEG V. VASILYEV, University of Colorado at Boulder, MATTIA GAZZOLA, PETROS KOUMOUTSAKOS, Swiss Federal Institute of Technology Zurich, Switzerland — In this talk we discuss preliminary results for the use of hybrid wavelet collocation – Brinkman penalization approach for shape optimization for drag reduction in flows past linked bodies. This optimization relies on Adaptive Wavelet Collocation Method along with the Brinkman penalization technique and the Covariance Matrix Adaptation Evolution Strategy (CMA-ES). Adaptive wavelet collocation method tackles the problem of efficiently resolving a fluid flow on a dynamically adaptive computational grid, while a level set approach is used to describe the body shape and the Brinkman volume penalization allows for an easy variation of flow geometry without requiring body-fitted meshes. We perform 2D simulations of linked bodies in order to investigate whether flat geometries are optimal for drag reduction. In order to accelerate the costly cost function evaluations we exploit the inherent parallelism of ES and we extend the CMA-ES implementation to a multi-host framework. This framework allows for an easy distribution of the cost function evaluations across several parallel architectures and it is not limited to only one computing facility. The resulting optimal shapes are geometrically consistent with the shapes that have been obtained in the pioneering wind tunnel experiments for drag reduction using Evolution Strategies by Ingo Rechenberg.
2:31PM CD.00008 Xwing: A 3D Viscous Design Tool for MAVs . DON GIPE, KAMRAN MOHSENI, University of Colorado at Boulder — Characterizing the 3D effects of low Reynolds number flow over low aspect ratio wings is computationally expensive using the full Navier-Stokes Equations. Recently, the use of Micro Aerial Vehicles as a tool for both scientific and military applications has grown leading to a diverse range of applications as well as designs. This development has created the need for a less computationally expensive approach for design of Micro Aerial Vehicles (MAVs). Xwing is a design tool for rapid calculation of aerodynamic forces of MAVs. Xwing uses an inviscid solver to compensate for the effect of tip vortices over a low aspect ratio wing. A potential flow – boundary layer matching technique was used at each 2D cross section using this effective angle of attack. The calculated displacement thickness at each cross section was used to obtain a new effective airfoil at each section. This iterative process could continue until convergence is achieved. The validity of the technique was tested in several cases including tapered, twisted, and swept wings at both low and high Reynolds number flows.

2:44PM CD.00009 A Poisson-Boltzmann solver on Non-Graded Adaptive Grid with Robin boundary conditions on Irregular Domains , ASDIS HELGADOTTIR, FREDERIC GIBOU, UCSB — We introduce a second-order solver for the Poisson-Boltzmann equation in arbitrary geometry in two and three spatial dimensions. The Poisson-Boltzmann equation can be used to represent the electric potential of a solution and is, therefore, of great interest in micro fluidics. The method introduced differs from existing methods solving the Poisson-Boltzmann equation in the two following ways: First, non-graded Quadtree (in two spatial dimensions) and Octree (in three spatial dimensions) grid structures are used; Quadtree/Octree grid structures save a significant amount of computational power at no sacrifice in accuracy. Second, Robin boundary conditions are enforced at the irregular domain’s boundary. The irregular domain is described implicitly and the grid does not need to conform to the domain’s boundary, which makes grid generation straightforward and robust. The resulting matrix is an M-matrix, thus the linear system is invertible, leading to a simple and robust second-order accurate solver.

Sunday, November 21, 2010 1:00PM - 3:10PM — Session CE Instability: Boundary Layers II — Long Beach Convention Center 102C

1:00PM CE.00001 Design of a DBD Plasma Actuator Array to Control Stationary Cross Flow Modes in a Supersonic Boundary Layer , CHAN-YONG SCHUELE, ERIC MATLIS, THOMAS CORKE, University of Notre Dame, STEPHEN WILKINSON, NASA Langley Research Center — The control of cross flow dominated laminar turbulent transition is crucial for the improvement of efficiency of supersonic aircraft. Passive methods such as distributed micron sized roughness elements have proven to work efficiently as laminar flow control devices in subsonic and as we could recently show in supersonic flows. This study describes the replacement of micron sized roughness elements with an array of dielectric barrier discharge (DBD) plasma actuators in order to excite less amplified stationary cross flow modes. These are intended to suppress the growth of the naturally occurring most amplified stationary modes. The use of DBD plasma actuators allows for a dynamic control that can respond to changing flight conditions, which is difficult to achieve with traditional roughness elements. Experiments have been performed in the 0.5 m Mach 3.5 NASA LaRC Supersonic Low Disturbance Tunnel on a 7° half angle sharp cone at a 4.3° angle of attack, and a unit Reynolds number of 250000/in.

1:13PM CE.00002 Hypersonic boundary-layer instability with localized roughness , OLAF MARXEN, GIANLUCA IACCARINO, ERIC SHAQFEH, Stanford University — Understanding the process of laminar-turbulent transition in supersonic flows has important implications for the design of thermal protections systems for hypersonic vehicles. A localized three-dimensional roughness element inside the boundary layer on the surface of such a vehicle may profoundly alter the instability of the boundary layer, and hence the transition process. However, our understanding of this alteration is far from comprehensive. A numerical investigation of a laminar flat-plate boundary layer with a localized 3-D roughness is carried out. The roughness height is on the order of half the boundary layer thickness. Streamwise vortices behind the roughness deform the boundary layer, leading to a strong low speed and high temperature streak behind the roughness. Moreover, wall-normal as well as spanwise gradients result from these streaks and are responsible for creating additional instabilities of the deformed boundary layer. Simulations have been performed for spanwise symmetric and asymmetric roughness shapes. Excitation of small-amplitude disturbance upstream of the roughness in the simulation allows to trigger the instabilities. Growth rates and amplitude functions will be compared for the different roughness shapes.

1:26PM CE.00003 Stability Analysis of a Mach 10 Boundary Layer with Nonequilibrium Chemistry , SHIRIN GAFFARI, OLAF MARXEN, GIANLUCA IACCARINO, ERIC SHAQFEH, Stanford University, TIERRY MAGIN, von Karman Institute — High temperature conditions in high Mach number flights can invalidate the assumption of a calorically perfect gas. As temperature rises, thermodynamic and transport properties of the gas mixture become not only a function of temperature but also of the chemical composition. If chemical nonequilibrium exists, additional transport equations for the species densities should be solved. Chemical nonequilibrium in the bulk can strongly affect boundary layer stability and transition to turbulence and thus it is an important capability to have in direct numerical simulation of high Mach number flows. At present, not many high-order numerical methods are capable of handling the high temperature regime. We examine boundary layer stability of a Mach 10 flow over a flat plate for the bulk in chemical nonequilibrium but thermal equilibrium. We carry out a high-order numerical integration of the Navier-Stokes equations via direct coupling to a library that computes gas properties based on the kinetic theory. Spatial amplification of small disturbances that may lead to transition on an isothermal or adiabatic flat plate are investigated.

1:39PM CE.00004 Boundary-layer transition on a flared cone in a Mach 6 quiet wind tunnel1 , JERROD HOFFERTH, WILLIAM SARIC, Texas A&M University — The Mach 6 Quiet Tunnel at Texas A&M is a low-disturbance blowdown facility suitable for boundary-layer stability and transition research. Following its reactivation in 2009, initial testing confirmed the presence of low-disturbance (< 0.1% \( P' / \overline{P} \)) freestream flow at select locations on the centerline of the nozzle for settling chamber pressures up to 10 atm, and a fully-traversed freestream flow-quality assessment is currently underway. As a third performance benchmark to complement these direct measurements, the present work measures the transition location on the NASA Langley 93-10 flared-cone model. This model has a 0.5m length, beginning as a 5° half-angle circular cone. At the \( X = 25.4 \text{mm} \) station, a flare of surface radius 2.35mm begins which is intended to induce transition within the quiet test core. Boundary-layer transition is detected on the thin-walled model by an observed surface temperature rise using an array of 51 embedded thermocouples. Transition data are presented for a sharp (2.5 \( \mu \text{m} \)) nose-tip radius case for comparison with the Lachowicz & Chokani (1996 data). Data for larger-radius nose-tips are also presented.

1 Work supported by AFOSR Grant FA9550-09-1-0341; AFOSR/NASA National Hypersonic Research Center in Laminar-Turbulent Transition.
1:52PM CE.00005 Hypersonic boundary layer instabilities affected by various porous surfaces\(^1\), XIAOWEN WANG, XIAOLIN ZHONG, University of California, Los Angeles — Hypersonic boundary layer instabilities of a Mach 5.92 flow over a flat plate affected by various porous surfaces are studied by numerical simulations. Steady base flow is obtained by solving compressible Navier-Stokes equations with a fifth-order shock-fitting method and a second-order TVD scheme. Stability simulations consist of two steps: (1) disturbances corresponding to a single boundary layer wave (mode F or mode S) are superimposed at a cross-section of the boundary layer near the leading edge to show spatial development of the wave; (2) porous coatings are used downstream of the superimposed waves to investigate its effect on boundary-layer instabilities. The results show that porous coating only has local effects on the instabilities of mode S and mode F. In porous region, Mack’s first mode is destabilized whereas Mack’s second mode and Mode F are stabilized. For felt-metal porous coating, destabilization of Mack’s first mode is so significant that disturbances are slightly destabilized when porous coating are put on the whole flat plate. At approximately the same porosity, regular structure porous coating is weaker in first mode destabilization and second mode stabilization than felt-metal porous coating.

\(^1\)Supported by the AFOSR/NASA National Center for Hyersonic Research in Laminar-Turbulent Transition.

2:05PM CE.00006 Investigation of a Turbulent Spot in a Hypersonic Cone Boundary Layer\(^1\), JAYAHAR SIVASUBRAMANIAN, HERMANN FASEL, University of Arizona — Direct Numerical Simulations (DNS) were performed to investigate the growth and breakdown of a localized disturbance into a turbulent spot in a sharp cone boundary layer at Mach 6. In order to model a natural transition scenario, the boundary layer was pulsed through a hole on the cone surface. The pulse disturbance developed into a three dimensional wave packet which consisted of a wide range of disturbance frequencies and wave numbers. The dominant waves within the resulting wave packet were identified as two dimensional second mode disturbance waves. In addition, weaker oblique waves were observed on the lateral sides of the wave packet. The developing wave packet grows linearly at first before reaching the nonlinear regime and eventually leads to localized patches of turbulent flow (turbulent spot). The wall pressure disturbance spectrum showed strong secondary peaks at the fundamental frequency for larger azimuthal wave numbers. This development indicates that fundamental resonance might be the dominant nonlinear mechanism for a cone boundary layer at Mach 6.

\(^1\)Funded by the AFOSR/NASA National Center for Hypersonic Laminar-Turbulent Transition Research and the AFOSR under grant FA9550-08-1-0211.

2:18PM CE.00007 Boundary-layer transition over aerodynamically-significant rotating bodies, STEPHEN GARRETT, ZAHIR HUSSAIN, ALISTAIR BARROW, PAUL TOWERS, University of Leicester — For practical reasons rotating-disk flow has served as the foremost model problem for studying transition in fully 3D incompressible boundary layers for over six decades and has a huge body of associated literature. However, continuing developments in spinning projectiles and aeroengines has led to the need to understand the onset of transition over rotating cones and spheroids as objects in their own right. Although numerous experimental observations have been published, these geometries received only little theoretical attention prior to 2002 when Garrett and co-workers commenced their work. In this paper we give a comparative study of the instability characteristics of the flows over these distinct geometries, discussing their similarities and differences with each other and the rotating-disk paradigm. The rotating-cone flow in particular is found to demonstrate significantly different characteristics as the half-angle is reduced below 40deg. This observation has led to the hypothesis and ultimate identification of an alternative instability mode which is expected to dominate for slender cones. Theoretical studies using numerical and asymptotic techniques are discussed. Comparisons are made to existing experimental data, and in many cases excellent agreement is observed for measurable properties. Where close agreement is not seen, we discuss possible reasons why.

2:31PM CE.00008 The centrifugal instability of the boundary layer on a slender rotating cone in a forced free-stream, ZAHIR HUSSAIN, STEPHEN GARRETT, University of Leicester, SHARON STEPHEN, University of Birmingham — The laminar-turbulent transition within the boundary layer over a slender rotating nose cone (for example a spinning missile) can lead to increases in drag, with negative implications for control and targeting. However, continuing developments on spinning projectiles, which have furthered understanding of the onset of laminar-turbulent transition over rotating cones, may lead to modifications and significant cost savings. Experiments in the literature have shown that increasing the incident free-stream has a stabilizing effect on these spiral vortices. Furthermore, Kobayashi (1981) has calculated the stability diagram for a slender cone of half-angle 15\(^\circ\) using the Orr-Sommerfeld approximation. In this study, we provide a new mathematical description of the onset of counter-rotating spiral vortices observed for a 15\(^\circ\) rotating cone placed in forced free-streams of varying strength. In particular, we resolve appropriate scalings in order to include variations in the basic-flow profiles, accounting for the influence of streamline curvature. A combined large Reynolds number and large vortex wavenumber analysis is used to obtain the asymptotic branch of neutral stability. Our results capture the effects of the governing centrifugal Görtler instability mechanism, and lead to favorable comparisons with existing numerical neutral stability curve results.

2:44PM CE.00009 Sidewall boundary layer instabilities in a rapidly rotating cylinder driven by a differentially co-rotating lid, JUAN LOPEZ, Arizona State University, FRANCISCO MARQUES, Universitat Politècnica de Catalunya — The flow in an enclosed completely filled rapidly rotating cylinder that is driven by the differential co-rotation of the lid top is studied numerically. Although the flow is in a very simple geometry, the fast background rotation and large differential rotation of the lid lead to very thin boundary layers with a variety of instability modes with very fine spatial scales as well as inertial waves that are sustained in the fast rotating interior flow and which interact with the viscous modes in the sidewall boundary layer leading to complex spatio-temporal dynamics. The numerical simulations are compared and contrasted to experimental visualizations of the sidewall boundary layer instabilities reported by Hart and Kittelman (1996), that include axisymmetric rolls propagating down the sidewall layer, backwards tilted diagonal rolls that precess slightly retrograde with respect to the rotating sidewall, forward tilted rolls with prograde precession significantly faster than the sidewall rotation, and a wavy turbulent state that has backwards tilted structures erupting from deep within the sidewall layer into the interior and are riding on the forward tilted diagonal rolls in the deep layer.

2:57PM CE.00010 Influence of a recent Transition Model on Complex Nonsteady Boundary Layer Flows with Dynamic Stall and Multiple Phases, ADAM LAVERY, MICHAEL KINZEL, GANESH VIJAYAKUMAR, JAMES BRASSEUR, ERIC PATERSOON, JULIE LINDAU, Penn State University — Computational fluid dynamics (CFD) simulations are prone to inaccuracies associated with incorrectly formulated physical models. Common in CFD is the spurious treatment as locally laminar flow regions as turbulent, resulting in incorrect turbulent-boundary-layer profiles, separated-flow behavior, and local skin-friction coefficients. The combined effects impacts global measures like drag, lift coefficient, and wake intensity. Recently, Menter & Langtry (AIAA 2009) developed a transition model applicable to unsteady three-dimensional CFD codes that shows promise to improve the prediction of local laminar regions. Our aim is to evaluate the accuracy of this model with the additional complexities of unsteady flow around rotating wind turbine blades and multiphase flows using codes designed within OpenFOAM. We investigate how transition and locally laminar flow regions impact various complex problems of interest including: (1) stationary S809 airfoil through stall, (2) an oscillating S809 airfoil in dynamic stall, and (3) a ventilated gaseous cavity in a liquid flow. We will evaluate the efficacy of the model by comparing with experimental results, and shall evaluate the impact on integral measures and flow details. Supported by NSF & DOE.
1:00PM CF.00001 Comparison of two-phase Darcy’s law with a thermodynamically consistent approach, STEFFEN BERG, Shell International Exploration & Production B.V., JENNIFER NIESSNER, University of Stuttgart, S. MAUD HASSANIZADEH, Utrecht University — The extended Darcy’s law is a commonly used description of immiscible two-phase flow in porous media. Fractional flow theory and reservoir engineering in the oil & gas industry is to a large extent based on this approach. In this description, the hydraulic conductivities of the porous medium for the two phases are parameterized with relative permeability-saturation functions which were introduced as empirical relationships. Within the last two decades, more advanced and physically based descriptions for multiphase flow in porous media have been developed. In this work, the extended Darcy’s law is compared to a thermodynamically consistent approach by Hassanizadeh and Gray (1990) which explicitly takes the important role of phase-interfaces into account, both as entities and as parameters. It turns out that the extended Darcy’s law and the thermodynamically based approach are compatible if either (i) relative permeabilities are a function of saturation only, but capillary pressure is a function of saturation and specific interfacial area or (ii) relative permeabilities are a function of saturation and saturation gradients. The latter would imply a more complex material behavior than commonly assumed in particular for the general case of irreversible displacement.

1:13PM CF.00002 Two Phase Flow in Porous Media with Dynamic Capillary Pressure1, KIMBERLY SPAYD, MICHAEL SHEARER, North Carolina State University — The one dimensional Buckley-Leverett equation for two phase flow in porous media is modified by including a dependence of capillary pressure on the rate of change of saturation. This model, due to Gray and Hassanizadeh, results in a nonlinear partial differential equation that supports traveling waves corresponding to undercompressive shocks. These waves, which also appear in driven thin liquid films, have the property that small disturbances pass through them from front to back. We present analytic results that are confirmed by numerical simulations of initial value problems.

1:26PM CF.00003 Stokes’ second problem for an Oldroyd-B fluid in a porous half space, BALRAM SUMAN, NAZISH HODA — A modified form of Darcy’s law is used to study the flow of Oldroyd-B fluids in a semi-infinite porous domain bounded by an oscillating plate. A close form analytical expression is obtained using Laplace transform, which suggests that depending upon the choice of natural and forced frequencies markedly distinct velocity fields can be observed. The fluid oscillates in time with a frequency which is same as the plate frequency. However, there exists a phase lag that increases with increasing distance from the plate. At a fixed distance from the plate, the phase-lag increases with increasing Reynolds number, Re = U/η/α3/ν, and Wissenberg number, Wi = λU2/ν, and decreasing viscosity ratio, β = λυ/λ#, where U is the amplitude of plate oscillation, K and φ are permeability and porosity of the porous medium, the kinematic viscosity, and λr and λ are retardation and relaxation times, respectively. This is consistent with our analytical prediction that the phase-lag is pronounced when Wi Re2 >> λ2. The phase lag also decreases with increasing Strouhal number, St = ν/ωU, where ω is the plate oscillation frequency, and it vanishes in the limit of St = 0. Furthermore, the fluid velocity increases with increasing β and Re, and decreasing distance from the plate. Away from the plate the fluid velocity shows a strong transient effect where the amplitude of oscillation either increases or decreases with time depending on Re, St, and β.

1:39PM CF.00004 Double diffusive miscible viscous fingering, M. MISHRA, Indian Institute of Technology Ropar, India, P.M.J. TREVELYAN, A. ALMARCHA, A. DE WIT, NLPC, Universite Libre de Bruxelles, Belgium — Miscible viscous fingering (VF) classically occurs when a less viscous fluid displaces a more viscous one in a porous medium. We analyze the influence on such VF of differential diffusion between two species each of them influencing the viscosity of the fluids at hand. We show that such double diffusive effects can destabilize the classically stable situation of a more viscous fluid displacing a less viscous one. On the basis of a time-dependent linear stability analysis, all possible instability scenarios are classified in a parameter space spanned by the log-mobility ratios of each species and by the ratio of diffusion coefficients. Numerical simulations of the full nonlinear problem confirm the existence of the predicted instability scenarios and highlight the influence of differential diffusion effects on the nonlinear fingering dynamics.

1:52PM CF.00005 Geometry-driven charge accumulation in electrokinetic flows through laminates, B.S. TILLEY, B. VERNESCU, J.D. PLUMMER, Worcester Polytechnic Institute — In the remediation of charged species from contaminated saturated soils, electric fields are applied to move the charge from the bulk toward the electrodes. Debye-layers occur at the soil/fluid interface which allows for advective transport of charge through electroosmosis, along with transport due to electrodiffusion (electrophoresis). However, depending on the valence of the species, these effects may act in an opposite or competing manner. We model the soil as an array of purely dielectric laminates with nonuniform thickness whose spatial variation in the streamwise direction occurs on a much longer length scale than the spacing between laminates (i.e. pore spacing). We derive an asymptotic model that incorporates lubrication pressures, dispersive effects in the electric field correction, and species equations for ion concentrations in the liquid. Electro-neutrality is not assumed in the fluid region. In the case of monovalent species, we find that spatial variations in the pore structure can lead to accumulation of charge where both the fringe electric field converges and the advective transport of charge is weak. Comparisons with results found in experiments are discussed.

2:05PM CF.00006 Capillary climb dynamics in the limits of prevailing capillary and gravity force, H.K. NAVAZ, B. MARKICEVIC, Kettering University, B. BIJELJIC, Department of Earth Science and Engineering, Imperial College — The dynamics of the capillary climb of a wetting liquid into a porous medium that is opposed by gravity force is studied numerically. The capillary network model, in which an actual porous medium is represented as a network of pores and throats, is used. The numerical results for the capillary climb reveal that there are at least two distinct flow regimes. The first regime is characterized by the capillary force being much larger than the gravity force. In this regime the Washburn solution can be used to predict the changes of climbing height over time. In the second regime the capillary and gravity forces become comparable, and one observes a slower, increased rate of climb that is height dependent. We model the soil as an array of purely dielectric laminates with nonuniform thickness whose spatial variation in the streamwise direction occurs on a much longer length scale than the spacing between laminates (i.e. pore spacing). We derive an asymptotic model that incorporates lubrication pressures, dispersive effects in the electric field correction, and species equations for ion concentrations in the liquid. Electro-neutrality is not assumed in the fluid region. In the case of monovalent species, we find that spatial variations in the pore structure can lead to accumulation of charge where both the fringe electric field converges and the advective transport of charge is weak. Comparisons with results found in experiments are discussed.

2:18PM CF.00007 The types of boundary conditions in the secondary capillary flow and liquid distribution, B. MARKICEVIC, H.K. NAVAZ, Kettering University — After depositing a wetting liquid onto a porous medium surface, and under the influence of the capillary pressure, the liquid is imbibed into porous medium creating a wetted imprint. The flow within the porous medium does not cease once all the liquid is imbibed, but it continues as a secondary capillary flow, where the liquid flows from large pores into small pores along the liquid interface. The flow is solved using the capillary network model, and influence of the boundary condition on the liquid distribution within porous medium is investigated. The porous medium boundaries can be defined as open or closed boundaries, where an open boundary is treated as a part of the liquid interface. In contrast, the closed boundary is defined as a static entity, in which the potential condition for flow to take place is never satisfied. By defining the porous medium boundaries as open or closed, it is possible to obtain a very accurate liquid distribution within the porous medium. The liquid saturation, phase flow direction, and fluid front evolution are highly sensitive to the boundary condition. One of the most popular boundary conditions in literature is the so-called constant capillary pressure boundary condition, which is widely applied to model the capillary flow in porous media. In this condition, the capillary pressure on the boundary is set to a constant value, which corresponds to the wetting or non-wetting state of the boundary. In this study, we investigated the influence of boundary conditions on the liquid distribution within porous medium to be controlled.
TAMAY ÖZGÖKMEN, University of Miami — The ocean’s surface mixed layer is notoriously complex due to high spatial and temporal gradients of density and
It can thus be readily adapted to structured, unstructured or coordinate mapping based models. Since the expansion is derived analytically for an arbitrary elliptic operator, it is independent on the particular choice of spatial discretization.
we show that if the full nonhydrostatic equations are needed, the expansion can be used in lieu of multi-phase flow in porous media will be based. Such a numerical framework consists of a h-adaptive refinement method, an entropy-based artificial
diffusive term, a new adaptive operator splitting method and efficient preconditioners. In particular, it is emphasized that we propose a new efficient adaptive operator splitting to avoid solving a time-consuming pressure-velocity part every saturation time step and, most importantly, we also provide a theoretically numerical analysis as well as proof. A few benchmarks will be demonstrated in the presentation.
2:57PM CF.00010 Pore-scale Modeling of Transport Phenomena in a Vanadium Redox Battery using X-ray Tomography and the Lattice Boltzmann Method, ABHIJIT JOSHI, E. CAGLAN KUMBUR, YING SUN, Drexel University — The Vanadium Redox Battery (VRB) promises to be an attractive option for storing electrical energy from renewable energy sources and delivering the stored energy to the grid whenever it is required. In this work, a novel methodology is proposed for modeling the transport mechanisms in the VRB including the flow of electrolyte, chemical species transport through the electrolyte and electrochemical reactions at active sites. The detailed geometry of the electrode is obtained using X-ray computed tomography (XCT) and this geometry is characterized to calculate porosity, pore-size distribution, connectivity and the active surface area. The processed XCT data is then used as geometry input for modeling transport processes in the VRB. The flow of electrolyte through the pore-space within the electrodes and the transport of ionic species in these pores is modelled using the lattice Boltzmann method (LBM). An electrochemical model based on the Butler-Volmer equation is used to provide species flux boundary conditions at the surface of the carbon fibers and to provide the necessary coupling to the local concentration of these species present in the pore space. Finally, model predictions for the steady-state discharge of the VRB are then compared with results reported in the literature.
Sunday, November 21, 2010 1:00PM - 3:10PM — Session CG Mini-Symposium on Computational Strategies for the Simulation of Nonlinear Waves and Turbulence in Environmental Flows Long Beach Convention Center 103B
1:00PM CG.00001 Direct Numerical Simulations of Stratified Turbulence, S.M. DE BRUYN KOOPS, Univ. of Massachusetts — In the 1990’s, computers became powerful enough to enable direct numerical simulations (DNS) of turbulent flows which were fully consistent with laboratory data. Prior to that, low Reynolds number effects encouraged the belief that simulation results must be validated against physical measurements. Today, many researchers accept DNS as complimentary to laboratory data for studying engineering turbulence. We are now at a comparable breakthrough point with DNS of stratified turbulence. Whereas simulations with strong stratification just a few years ago could be run with a buoyancy Reynolds number, $Re_b$ of about 50, it is now practical to simulate flows with $Re_b > 500$, well beyond the value accepted for being relevant to the ocean or atmosphere. Statistics from DNS of forced homogeneous stratified turbulence run with 8.6 billion grid points are reported for a wide range of Froude numbers. Spectral energy balances are shown, as are the probability density functions of turbulent quantities. Preliminary results for runs with 69 billion points are also presented, along with an invitation for collaborative uses of these data.
1:26PM CG.00002 Hydrostatic ma non troppo. Leptic expansion and Poisson solvers in thin domains1, A. SCOTTI, U. North Carolina-Chapel Hill — The hydrostatic approximation of internal waves fails when the horizontal scale is comparable to the local depth. The resulting dispersion opposes nonleucopering. Theoretical models rely on approximations to achieve physically reasonable dispersion while ocean models rely on fully nonhydrostatic equations, whose solution involves expensive Poisson solvers. We show that even within numerical models there exists a continuum between hydrostatic and fully nonhydrostatic behavior. A formal expansion of the solution of the Poisson problem can be used to explore the grey area separating hydrostatic from nonhydrostatic scales. We show that an asymptotically correct amount of dispersion can be added to internal waves without incurring the full cost of a nonhydrostatic solution. We also show that if the full nonhydrostatic equations are needed, the expansion can be used in lieu of a preconditioner. Since the expansion is derived analytically for an arbitrary elliptic operator, it is independent on the particular choice of spatial discretization. It can thus be readily adapted to structured, unstructured or coordinate mapping based models.
1 In collaboration with E. Santilli and J. Wilson, University of North Carolina-Chapel Hill.
1:52PM CG.00003 On the Interaction of Buoyant Plumes With Ocean Mixed-Layer Fronts1, TAMAY ÖZGÖKMEN, University of Miami — The ocean’s surface mixed layer is notoriously complex due to high spatial and temporal gradients of density and velocity fields. The understanding and modeling of such flows have a wide range of applications. For instance, anomalous currents and density perturbations in the acoustic and optical environment can affect a variety of naval operations. These flows can also influence strongly the dispersion of surface and sub-surface pollutants. Large eddy simulations of an idealized mixed-layer problem are conducted using the spectral element model Nek5000. Sampling strategies of these fields are investigated using passive tracers and Lagrangian particles. These idealized fields are then used in order to explore the behavior of a buoyant plume through the water column, namely its surface and sub-surface dispersion, which is motivated by the Deepwater Horizon oil spill.
1 In collaboration with Paul Fischer, Argonne National Laboratory.
2:18PM CG.00004 Computational Investigations of Gravity and Turbidity Currents

ECKART MEIBURG, University of California, Santa Barbara — We will present an overview of high-resolution, Navier-Stokes based simulations of gravity and turbidity currents. The turbulence currents we consider are driven by particles that have negligible inertia and are much smaller than the smallest length scales of the buoyancy-induced fluid motion. For their mathematical description an Eulerian approach is employed, with a transport equation for the particle-number density. The governing equations are integrated numerically with high-order, compact finite difference techniques for rectangular geometries, and with second order finite difference methods for complex geometries. Arbitrary seafloor topographies are implemented via an immersed boundary method. We will discuss differences between two- and three-dimensional turbidity current dynamics in the lock-exchange configuration, and we will introduce some effects due to complex topography. Results will be shown regarding non-Boussinesq effects, and the unsteady interaction of a gravity current with a submarine structure, such as a pipeline. Furthermore, we will briefly discuss the linear stability problem of channel and sediment wave formation by turbidity currents.

1 In collaboration with M. Nasr, University of California, Santa Barbara.

2:44PM CG.00005 High resolution simulations of turbulent flows in open-channels

THORSTEN STOESSER, Georgia Institute of Technology — The results of recent high resolution large-eddy simulations (LES) of selected turbulent open channel flows are presented and discussed. An efficient, finite volume based Navier-Stokes solver is employed to simulate the flow over a rough bed and through idealized emergent vegetation. The fluid phase is solved entirely on a Cartesian grid and different immersed boundary methods (IMB) are used to account for the solid, non-Cartesian roughness elements. First and second order statistics are compared to measurements and the accuracy of the different IMBs is assessed. The frequency of turbulence events (like sweeps, ejections) and their contribution to Reynolds stresses is quantified and the anisotropy of turbulence is described. Based on the turbulence statistics, the utilization of structures eduction methods and aided by visualizations and animations the generation and fate of the dominating coherent structures in the flows is discussed.

1 In collaboration with S.B. Bomminayuni, Georgia Institute of Technology.

Sunday, November 21, 2010 1:00PM - 3:10PM — Session CH Convection and Buoyancy Driven Flows II Long Beach Convention Center 103C

1:00PM CH.00001 Spatially Localized Binary Fluid Convection

EDGAR KNOBLOCH, University of California at Berkeley, DAVID LO JACONO, ALAIN BERGEON, IMFT and UPS, Toulouse — Three-dimensional convection in a binary mixture in a porous medium heated from below is studied. For negative separation ratios steady convection patterns, spatially localized in one or two dimensions, are computed and numerical continuation is used to examine the growth, stability and proliferation of each pattern as parameters are varied. The results are complemented by direct numerical simulations with periodic boundary conditions in the horizontal.


1:13PM CH.00002 Evolution of Double-Diffusive Convection in Low-Aspect Ratio Containers

SUHAS POL, Los Alamos National Labs, HARINDRA FERNANDO, University of Notre Dame, STEPHEN WEBB, Sandia National Labs — Laboratory experiments and phenomenological modeling were undertaken to investigate the influence of container sidewalls on the evolution of diffusive layering in confined double-diffusive systems. Such flow configurations are common in engineering situations, including underground storage caverns of national strategic petroleum reserves. The laboratory flow configuration consisted of a linearly salt stratified fluid subjected to either heating from below or uniform heating from both the bottom and sidewalls. The growth of the mixed layers separated by diffusive interfaces was monitored using PIV and traversing temperature/conductivity probe techniques. The importance of aspect ratio effects was inferred from the bottom-layer growth measurements, which undergoes a transition upon onset of side-wall effects (aspect ratio ~ 1). A second transition was noticed at an aspect ratio ~ 2 when elongated eddies break down in to smaller sizes. The combined side and bottom wall heating case was strikingly different from the bottom heating case, wherein layers of approximately equal heights are generated rather rapidly in the former as a result of convective plumes rising along the sidewalls and their arrest by the background stable density gradient. Theoretical arguments were advanced to explain and parameterize experimental observations.

1:26PM CH.00003 Turbulent Rayleigh-Bénard convection in containers with rectangular wall roughness elements

OLGA SHISHKINA, CLAUS WAGNER, DLR - Institute for Aerodynamics and Flow Technology — The work is devoted to numerical investigation of the heat transport in turbulent Rayleigh-Bénard convection enhanced by roughness of the heated/cooled plates. The roughness is determined by rectangular obstacles, which are located at the plates and are heated/cooled in the same way as the corresponding plates. Based on the obtained numerical results a theoretical model to predict the heat transport (Nusselt number) in natural convection in enclosed domains with heated/cooled obstacles is developed.

1:39PM CH.00004 Convective Heat Transfer in Bulk- and Boundary-Dominated Regimes in Turbulent Thermal Convection

PING WEI, RUI NI, XIAO-ZHENG ZHAO, KE-QING XIA, The Chinese University of Hong Kong — We report Nusselt number measurements in Rayleigh-Bénard convection systems with modified boundary conditions and over the range of the Rayleigh number (Ra) spanning from \(3 \times 10^5\) to \(8 \times 10^7\) and with the Prandtl number \(Pr \sim 4.3\). These measurements were made in three convection cells: (1) both the top and bottom plates of the cell have flat smooth surface; (2) the top plate has a flat smooth surface while the bottom plate has a rough surface in the form of regularly-arrayed pyramids; and (3) the top plate is rough as in (2) but the bottom plate is smooth. All these cells have cylindrical shape with aspect ratio one. The experimental results suggest that the \(Nu \sim Ra\) relationship can be represented by the combination of two power laws, corresponding to the bulk-dominant regime (exponent = 1/2) and boundary layer dominant one (exponent =1/4) of the Grossmann-Lohse model. The behaviors of the coefficients of the two power laws suggest that the roughness of the plate can enhance the contribution of bulk and push the system to change from the boundary dominant state to bulk dominant state.

1 Work supported by the Research Grants Council of Hong Kong SAR (Project Nos. CUHK 403806).
indirectly confirmed by temperature measurements along the sidewall of the convection cell. caused by the weaker, elliptical LSC, allowing Ekman pumping to take over heat transport more easily. The disappearance of the LSC around Ro model that explains the existence of a bifurcation at finite 1 inverse Rossby number found transitions between different turbulent states are rare, since turbulence is expected to sample all of phase space over wide parameter ranges. However, it was found that a cylindrical systems, vorticies that extract additional fluid out of the thermal boundary layers at the sample top and bottom. As found in experiments and numerical simulations in LSC in a cylindrical cell with aspect ratio 2. The flow patterns and the heat transfer rate for different Rayleigh numbers. The natural convection analysis is carried out from the subcritical steady state (pure axially downwards (ii) a homogeneous internal heat source in the fluid, and (iii) a uniform low temperature at the wall of the sphere. We show the influence of heat source variations greatly: some of the orbits belong to different branches and a global bifurcation is suspected of delimiting the lower limit of periodic states.

The flow patterns and the heat transfer rates for the considered Rayleigh numbers (0 < Ru < 3 × 10^7) and Taylor numbers (0 < Tu < 1 × 10^8) are in satisfactory agreement with results previously published in the literature.

The flow patterns and the heat transfer rates for different Rayleigh numbers. The natural convection analysis is carried out from the subcritical steady state (pure conduction) regime to the supercritical non-steady state regime. The mesh based Spectral Element method (SEM) has been used to solve the fluid equations in a Cartesian coordinate system in a rotating reference frame. The fluid sphere is discretized by using non-regular hexahedra with straight sides macro elements. Using this approach the singularity that appears at the poles of the sphere, when the governing equations are formulated in the spherical coordinate system, is avoided. The flow patterns and the heat transfer rates for the considered Rayleigh numbers (0 < Ru < 3 × 10^7) and Taylor numbers (0 < Tu < 1 × 10^8) are in satisfactory agreement with results previously published in the literature.

Supported by NSF Grant DMR07-02111.


2:57PM CH.00010 Rotating Rayleigh-Bénard convection in a cylindrical cell with aspect ratio two. JIM V. OVERKAMP, Eindhoven University of Technology, RICHARD J.A.M. STEVENS, DETLEF LOHSE, University Twente, HERMAN J.H. CLERCX, Eindhoven University of Technology — Turbulent thermal convection occurs for example in the Earth’s atmosphere and in the Earth’s liquid outer core, which can be represented by the rotating Rayleigh-Bénard problem. We investigate the effect of rotation on heat transport and the presence of the large-scale circulation (LSC) in a cylindrical cell with aspect ratio Γ = 2. The Rayleigh number Ra in these experiments is varied between 2 × 10^8 and 1.5 × 10^9, with Prandtl number Pr ≈ 4.4. Non-rotating measurements of heat transport agree with literature data (both for cells with Γ = 1 and 2). Heat transport in rotating Rayleigh-Bénard convection (rotation vector parallel with the cylinder axis) is increased by up to 20% (Ra = 3 × 10^9), which is larger than reported heat transfer enhancements for Γ = 1 cells (and similar Ra). The onset of this enhancement by rotation is located at a higher Rossby value than for the cell with Γ = 1. This is likely caused by the weaker, elliptical LSC, allowing Ekman pumping to take over heat transport more easily. The disappearance of the LSC around Ro = 7 is also indirectly confirmed by temperature measurements along the sidewall of the convection cell.

Sunday, November 21, 2010 1:00PM - 3:10PM – Session CJ Flow Control II Long Beach Convention Center 201A
1:00PM CJ.00001 Feedback Flow Control of a Periodically Pitching Airfoil. ANDREW LOMBARDI, PATRICK BOWLES, THOMAS CORKE, ERIC MALIUS, University of Notre Dame — A method for detecting incipient flow separation using plasma actuators is presented. The detection scheme relies upon the receptivity of the flow to unsteady disturbances that are introduced by the flow actuator. The receptivity to the unsteady disturbances is heightened as the flow approaches the separation limit, and subsequently can be detected downstream. This is demonstrated on a dynamically pitching airfoil that progresses through a dynamic stall cycle. A plasma actuator is located at the leading edge and pulsed at a frequency that is optimal to re-attach the flow. A pressure sensor monitors the unsteady pressure disturbances on the suction-side of the airfoil. Short-time Fourier analysis is used to capture the time-frequency behavior of the pressure sensor time series. Simultaneous flow visualization using a high-speed camera aid in elucidating the fluid response to the actuator input. The method not only provides a precursor for flow separation, but also an indicator when conditions exist where active re-attachment control is no longer needed. A closed-loop, feedback control scheme based on this is demonstrated.

1:13PM CJ.00002 Aerodynamic Flow Control using Distributed Active Bleed. JOHN M. KEARNY, ARI GLEZER, Georgia Institute of Technology — The aerodynamic effects of large-area air bleed that is driven through surface openings by pressure differences across a lifting airfoil and regulated by addressable, arrays of integrated louvers have been investigated in wind tunnel experiments. Time-dependent interactions between the bleed and cross flows alter the apparent aerodynamic shape of the lifting surface and consequently the distributions of aerodynamic forces and moments. The lift and pitching moment can be significantly altered over a wide range of angles of attack from pre- to post-stall by independently-controlled bleed near the leading (LE) and trailing (TE) edges. While TE bleed effects nearly-linear variation of the pitching moment with minimal changes in lift, LE bleed leads to large variations in lift and pitching moment with minimal drag penalty. Phase-locked PIV shows the effects of the bleed on the flow on the suction surface and in the near wake. Supported by AFOSR.

1:26PM CJ.00003 Numerical Simulations of Natural and Actuated Flow over a 3D, Low-Aspect-Ratio Airfoil. GUILLAUME BRES, Exa Corporation, DAVID WILLIAMS, Illinois Institute of Technology, TIM COLONIUS, California Institute of Technology — Numerical simulations of the unsteady flow over a low-aspect-ratio, low Reynolds number semi-circular planform wing are performed using Lattice Boltzmann method. The simulations exactly match the flow conditions and the detailed geometry from previous wind-tunnel experiments, including the flow actuators installed internally along the leading edge of the wing. To reproduce the pulsed-blowing actuation used in the experiment, a single pulsed square wave forcing is imposed in the simulations as a mass flow boundary condition in the actuators. Three angles of attack, with the active flow control both on and off, are investigated. For both mean and unsteady lift and drag, the numerical simulations show good agreement with the experiments. In particular, the transient increase in lift after the forcing is turned off is well captured in the simulations. Both PIV measurements and transient numerical results indicate that this behavior is associated with the advection of large vortical structures generated by the flow actuation at the leading edge.

1:39PM CJ.00004 Aerodynamic Flow Control of a Maneuvering Airfoil. DANIEL P. BRZOZOWSKI, JOHN CULP, ARI GLEZER, Georgia Institute of Technology — The unsteady aerodynamic forces and moments on a maneuvering, free-moving airfoil are varied in wind tunnel experiments by controlling vorticity generation/accumulation near the surface using hybrid synthetic jet actuators. The dynamic characteristics of the airfoil that is mounted on a 2-DOF traverse are controlled using position and attitude feedback loops that are actuated by servo motors. Bi-directional changes in the pitching moment are induced using controllable trapped vorticity concentrations on the suction and pressure surfaces near the trailing edge. The dynamic coupling between the actuation and the time-dependent flow field is characterized using simultaneous force and velocity measurements that are taken phase-locked to the commanded actuation waveform. The time scales associated with the actuation process is determined from PIV measurements of vorticity flux downstream of the trailing edge. Circulation time history shows that the entire flow over the airfoil readjusts within about 1.5 $T_{CONV}$, which is about two orders of magnitude shorter than the characteristic time associated with the controlled maneuver of the wind tunnel model. This illustrates that flow-control actuation can be typically effected on time scales commensurate with the flow’s convective time scale, and that the maneuver response is only limited by the inertia of the platform. Supported by AFOSR.

1:52PM CJ.00005 Active Flow Control on a Low Reynolds Number Wing. MATTHEW MUNSON, MORTEZA GHARIB, California Institute of Technology — Control of vortex formation has been shown to be a critical mechanism in some forms of animal flight. Flapping motions create advantageous flow structures which play a role in enhancing lift and increasing maneuverability. Active flow control may be capable of providing similar influence over vortex formation processes in fixed wing flight at small Reynolds numbers. Steady and pulsed mass injection strategies through simple slot actuators are used to explore the open-loop response of the flow around a simple low-aspect ratio wing. Flow dynamics and vortex formation will be quantitatively visualized with DPIV and flow forces will be simultaneously measured with a six-component balance.

2:05PM CJ.00006 Low Reynolds Number Flow Dynamics of a Thin Airfoil with an Actuated Leading Edge using Direct Numerical Simulation. KEVIN DROSST, SOURABH APTE, Oregon State University — Direct numerical simulations are performed to investigate the effect of a movable leading edge on the unsteady flow at high angles of attack over a flat, thin airfoil at Reynolds number of 14700 based on the chord length. The leading edge of the airfoil is hinged at one-third chord length allowing dynamic variations in the effective angle of attack through specified oscillations (or flapping). A fictitious-domain based finite volume approach ([Apte et al. (JCP 2009)]) is used to compute the flow over an airfoil with a flapping leading edge on a fixed background mesh. Cases were run at 20 degrees angle of attack to study the drag and lift characteristics with sinusoidal flapping of the leading edge about the hinge over a range of reduced frequencies ($k = r f_c U_\infty$). It is shown that high-frequency low amplitude actuation of the leading edge significantly alters the leading edge boundary-layer and vortex shedding and increases the mean lift-to-drag ratio. The concept of an actuated leading-edge flap has potential for development of control techniques to stabilize and maneuver low-Reynolds number micro-air vehicles in response to unsteady perturbations.

---

1 Sourabh Apte acknowledges ASEEE-SFFP Summer fellowship (2010) at Wright-Patterson Air Force Laboratory, Dayton, OH.

2 Supported by ONR.
2:31PM CJ.00008 Experimental investigations on the drag reduction mechanism of outer-layer vertical blades array1, INWON LEE, ASERC, Pusan National University, NAM HYUN AN, Division of Shipbuilding and Marine Eng., Koje College, KWING-SO CHOI, University of Nottingham, HO HWAN CHUN, ASERC, Pusan National University — An experimental assessment has been made of the drag reducing efficiency of the outer-layer vertical blades, which were first devised by Hutchins (2003). The local skin friction reduction of the blades was reported to reach as much as 30%. In the present study, a series of drag force measurements in towing tank have been performed toward the assessments of the total drag reduction efficiency of the outer-layer vertical blades. A maximum 9.6% of reduction of total drag was achieved. The scale of blade geometry is found to be weakly correlated with outer variable of boundary layer. In addition, detailed flow field measurements have been performed using time resolved PIV with a view to enabling the identification of drag reduction mechanism. The comparison of real-time turbulence structure observed in the xz-plane in the outer layer revealed that the blades array disturbs the growth of the large-scale structures in the spanwise direction.

1This research was sponsored by the ERC program (ASERC) of Ministry of Education, Science and Technology & National Research Foundation of Korea.

2:44PM CJ.00009 Benefits of Active Flow Control for Wind Turbine Blades. GUANNAN WANG, BASMAN ELHADIDI, JAKUB WALCZAK, Syracuse University, MARK GLAUSER, Syracuse University, HIROSHI HIGUCHI, Syracuse University — In this talk, the blade element momentum model is used to design a wind turbine and examine the benefit of active flow control. The results suggest that either the overall operational range of the wind turbine could be effectively enlarged by 80% with the same rated power output or the rated output power could be increased by 20% while maintaining the same level of operational range when the control is on. The optimal location for the actuator is found to be on the outboard of the blade beyond half of the radius. In light of these encouraging results and based on our earlier NACA 4412 flow control studies, a characteristic airfoil (e.g. DU-96-W-180) is being tested in a new anechoic wind tunnel facility at Syracuse University to determine the airfoil lift and drag characteristics with appropriate flow control while exposed to large scale flow unsteadiness. In addition, the effects of flow controllers on the noise spectrum of the wind turbine will also be assessed and measured in the anechoic chamber.

2:57PM CJ.00010 Characterizing rotor stator interaction (RSI) using CFD and experimentally obtained wake flow fields. MORTEN KJELDSEN, PAL H.E. FINSTAD, Norwegian University of Science and Technology, ROGER E.A. ARNDT, University of Minnesota — RSI is a major reason for noise and vibration, and reduced performance of turbomachinery. The stationary cascade upstream of the impeller stage is a source of variations in velocity due to angular momentum transfer, creating a cascade blade-to-blade variation. In addition a number of secondary flow fields due to boundary layer dynamics, such as wake flows, emerge from the cascade. At UMN a number of TR PIV fields have been captured downstream of a hydrofoil in liquid water, c=81mm and Re=5 (to 8)e5, for different AoAs and for selected passive flow control techniques. The wake trailing the foil is characterized by swirling structures, albeit far from regular shedding. One line of analysis of the captured wake flow fields has been to characterize the structures by a statistical averaged energy analysis over the structures. A second approach has been to use the experimentally obtained data as input in CFD analysis of the impingement of the wake on a rotating vane. Both the procedure and results are described.

Sunday, November 21, 2010 1:00PM - 2:57PM –
Session CK Free Surface Flows: Macro-Scale Phenomena
Long Beach Convention Center 201B

1:00PM CK.00001 Dragging a floating horizontal cylinder. DUCK-GYU LEE, HO-YOUNG KIM, Seoul National University — A cylinder immersed in a fluid stream experiences a drag, and it is well known that the drag coefficient is a function of the Reynolds number only. Here we study the force exerted on a long horizontal cylinder that is dragged perpendicular to its axis while floating on an air-water interface with a high Reynolds number. In addition to the flow-induced drag, the floating body is subjected to capillary forces along the contact line where the three phases of liquid/solid/gas meet. We first theoretically predict the meniscus profile around the horizontally moving cylinder assuming the potential flow, and show that the profile is in good agreement with that obtained experimentally. Then we compare our theoretical predictions and experimental measurement results for the drag coefficient of a floating horizontal cylinder that is given by a function of the Weber number and the Bond number. This study can help us to understand the horizontal motion of partially submerged objects at air-liquid interface, such as semi-aquatic insects and marine plants.

1:13PM CK.00002 The submarine’s wake. ADRIEN BENUMIGLIO, MARRIE LE MERRER, CHRISTOPHE CLANET, LadinX, Ecole Polytechnique — An object moving under water creates a wake at the surface. We measure the amplitude of the wake and the additive drag it creates on a sphere moving horizontally at different depths and speeds. We isolate two different comportements at high and low depth. Can we deduce the size and depth of a submarine from it’s wake?

1:26PM CK.00003 Experimental Investigation of Slaming Loads on a Flat Plate. DAVID JEON, California Institute of Technology, FRANCISCO HUERA-HUARTE, Universitat Rovira I Virgili, MATHIEU FU, MORY GHARIB, California Institute of Technology — Slaming loads on marine structures and vessels pose a hazard, with the potential for structural failure from the high momentary loads. With interest in higher speed vessels, exterior panels face tremendous impact loads from both wave impact and slamming. We have designed an experimental apparatus that can slam a variety of objects into a free surface at a range of deadrise angles and impact speeds. This system is instrumented with load cells to give us the force history of the impact, rather than the pressure on the face of the panel. Impact speeds over 5 m/s have been tested, with impact angles ranging from 1-25 degrees, using a foam core composite panel. We have documented the cushioning effect of trapped air between the plate and the free surface at small impact angles. We have also seen a correlation between the impact duration and the total force. The authors would like to thank the Office of Naval Research for their support of this experiment through award number N00014-06-1-O730. In addition, FJHH would like to acknowledge the support given by the European Commission through the Marie Curie IOF for actions for individuals (PIOF-GA-2008-219429).

1:39PM CK.00004 A comparison of model-scale experimental measurements and computational predictions for a large transom-stern wave. THOMAS T. O’SHEA, KRISTY L.C. BEALE, KYLIE A. BRUCKER, DONALD C. WYATT, SAIC, DAVID DRAZEN, ANNE M. FULLERTON, TOM C. FU, NSWCCD, DOUGLAS G. DOMMERMUTH, SAIC — Numerical Flow Analysis (NFA) predictions of the flow around a transom stern hull form are compared to laboratory measurements collected at NSWCCD. The simulations are two-phase, three-dimensional, and unsteady. Each required 1.15 billion grid cells and 200,000 CPU hours to accurately resolve the unsteady flow and obtain a sufficient statistical ensemble size. Two speeds, 7 and 8 knots, are compared. The 7 knots case is a dry transom condition. The results of a detailed comparison of the mean free surface elevation, surface roughness (RMS), and spectra of the breaking stern-waves, measured by Light Detection And Ranging (LiDAR) and Quantitative Visualization (QViz) sensors, are presented. All of the comparisons showed excellent agreement. The concept of height-function processing is introduced, and the application of this type of processing to the simulation data shows a k−5/3 power law behavior for both the 7 and 8 knot cases. The simulations also showed that a multiphase shear layer forms in the rooster-tail region and that its thickness depends on the Froude number.
1:52PM CK.00005 Computation of two-dimensional standing water waves. JON WILKENING, JIA YU, CHRIS RYCROFT, UC Berkeley — We develop a quasi-Newton trust-region shooting algorithm for solving two-point boundary value problems governed by nonlinear PDEs. We use our method to compute families of (time-periodic) standing water waves in two dimensions. To evolve the water wave in time, we use a spectrally accurate boundary integral collocation method. As a starting guess, we use analytically determined time-periodic solutions of the linearized problem about a flat surface. We then use our numerical method to continue these solutions beyond the realm of linear theory to explore the topology and bifurcation structure of a two-parameter family of standing waves (with mean depth and wave amplitude as parameters). Preliminary results suggest that if limiting wave profiles exist, they have more complicated singularities than the 90 degree angles previously conjectured.

2:05PM CK.00006 Computation of three-dimensional standing water waves. CHRIS RYCROFT, JON WILKENING, UC Berkeley — We develop a method for computing three-dimensional gravity-driven water waves, which we use to search for time-periodic standing wave solutions. We simulate an inviscid, irrotational, incompressible fluid bounded below by a flat wall, and above by an evolving free surface. The computations make use of spectral derivatives on the surface, but also require computing a velocity potential in the bulk, which we carry out using a finite element method with fourth order elements that are curved to match the free surface — this computationally expensive step is solved using a parallel multigrid algorithm which we have developed. We search for time-periodic solutions using the trust-region shooting method that was previously used to find two-dimensional standing water waves.

2:18PM CK.00007 Energy Transfer into Cross-Waves by Wavemakers. TATYANA KRASNOPOLSKAYA, Department of Vortex Motion, Institute of Hydromechanics NASU, VIATCHESLAV MELESHKO, Department of Theoretical and Applied Mechanics, Faculty of Mechanics and Mathematics, Kiev National Taras Shevchenko University, VYACHESLAV SPEKTOR, Department of Vortex Motion, Institute of Hydromechanics NASU — In the long channels, the cross-waves derive their energy directly from wavemaker not only from basic flow. In the present talk we show how cross-waves are generated in long rectangular channels by wavemaker even without having to take into account the presence of any basic flow waves. Here we apply Lamé's method of superposition for the first time in such channel geometry. This method allows one to construct a simple mathematical model, which shows how the cross-waves can be generated directly by the wavemaker motion. This mathematical model of the resonant cross-wave excitation is the easiest way to study pattern formation on fluid free surface. Our experimental observations agree with the theoretical results.

2:31PM CK.00008 Experimental Study of Free Surface Magnetohydrodynamic Flow. J. RHoads, E. ELDuNd, P. SLOBODA, E. SPENCE, H. JI, PPPL — Free surface MHD flows contain many interesting phenomena due to the interplay between the free boundary condition at the surface and the effects of the external magnetic field. This interaction can produce features distinct from other types of flow. The Liquid Metal Experiment (LMI) is designed to investigate the effects of a strong magnetic field applied orthogonal to the flow direction of an electrically conducting fluid. In order to study heat transfer under these conditions, a resistive heater and an infrared camera have been installed. Changes in the vortex street from the cylindrical heater have been observed as the field is increased. Additionally, the modification of underlying turbulent structures can be tracked using two position-sensitive diodes. This diagnostic records fluctuations of the surface from which the k-spectra can be extracted by using cross-correlation techniques. Lastly, a local velocity diagnostic is under development which should have the capability of mapping the velocity profile as a function of the magnetic field. An overview of the experiment and preliminary results will be presented.

2:44PM CK.00009 Experimental investigation of orbitally shaken bioreactor hydrodynamics. MARTINO RECLARI, MATTHIEU DREYER, MOHAMED FARHAT, Laboratory for Hydraulic Machines - EPFL — The growing interest in the use of orbitally shaken bioreactors for mammalian cell cultivation raises challenging hydrodynamic issues. Optimizations of mixing and oxygenation, as well as similarity relations between different culture scales are still lacking. In the present study, we investigated the relation between the shape of the free surface, the mixing process and the velocity fields, using specific imaging process of high speed visualization and Laser Doppler velocimetry. Moreover, similarity parameters were identified for scale-up purposes.

Sunday, November 21, 2010 1:00PM - 3:10PM — Session CL Biofluids: Physiological Feeding Long Beach Convention Center 202A

1:00PM CL.00001 Special functions of valve organs of blood-sucking female mosquitoes1. BOHEUM KIM, SANGJOON LEE — Food-feeding insects usually have valve organs to regulate the sucking flow effectively. Female mosquitoes sucking lots of blood instantaneously have a unique valve system between two pumping organs located in their head. The valve system seems to prevent reverse flow and to grind granule particles such as red blood cells. To understand the functional characteristics of this valve organ in detail, the volumetric flow rate passing through the valves and their interaction with the two-pumps need to be investigated. However, it is very difficult to observe the dynamic behaviors of pumping organs and valve system. In this study, the dynamic motions of valve organs of blood-sucking female mosquitoes were observed under in vivo condition using synchrotron X-ray micro imaging technique. X-ray micro computed tomography was also employed to examine the three-dimensional internal structure of the blood pumping system including valve organs.

1:13PM CL.00002 The ultrafast of an aquatic carnivorous plant. PHILIPPE MARMOTTANT, OLIVIER VINCENT, MARC JOYEUX, CATHERINE QUILLIET, Lab. Spectrometrie Physique, CNRS and University of Grenoble, SIMON POPPINGA, CARMEN WEISSKOPF, TOM MASSELMER, THOMAS SPECK, Botanical Garden of the University of Freiburg, Germany — Aquatic carnivorous bladderworts (Utricularia spp.) are plants that catch prey animals with suction traps. Here we present an experimental study with high speed video analyses of the extremely fast trapping movements, and show that suction is performed in less than a millisecond, much faster than previously thought. We reveal how the convex door morphology is optimized for a fast opening and closure, which we confirm by numerical simulations: the trapdoor is an elastic valve that buckles inside (entailing rapid opening) and then unbuckles (entailing rapid closure). These precise and reproducible motions are coupled with a strong suction swirl causing accelerations of up to 600 g, and leaving little escape chances for prey animals.

1:26PM CL.00003 Optimality and universal scaling for osmotically driven translocation of sugars in plants. TOMAS BOHR, Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, KAARE HARTVIG JENSEN, HENRIK BRUUS, Department of Micro- and Nanotechnology, Technical University of Denmark, JINKEE LEE, Division of Engineering, Brown University, NICOL AEZ ZWIERNECKI, Anonymous Arboretum, Harvard University, NOEL MICHELE HOLBROOK, Laboratory for Agricultural Machines - EPFL — The growth of plants depends on efficient translocation of sugars. The current belief is that this takes place predominantly through osmotically driven flow, passively generated by differences in sugar concentrations (the so-called Münch mechanism). We show that optimization of translocation speed predicts a universal scaling between the width of the conduits (phloem cells), the length of the plant and the length of the “loading zones” (the leaves). This unexpected scaling is verified by data from plants over several orders of magnitude is size, from small green plants to large trees.
1:52PM CL.00005 Microscopic filter feeders at an angle to nearby boundaries: Feeding restrictions and strategies, RACHEL PEPPER, University of Colorado, MARCUS ROPER, University of California Berkeley, SANGJIN RYU, Brown University, PAUL MATSUDIARA, National University of Singapore, HOWARD STONE, Princeton University — Microscopic sessile filter feeders are an important part of aquatic ecosystems and form a vital link in the transfer of carbon and nutrient into marine food webs. These filter feeders live attached to boundaries, consume bacteria and small detritus, and are in turn eaten by larger organisms. Such filter feeders survive by creating a feeding current that draws fluid towards them, and from which they filter their food of interest. Eddies form near these organisms as a result of fluid forcing near a boundary. The extent of these eddies, and their effect on the nutrient uptake of the organism, depend on the angle of fluid forcing relative to the boundary. For a model with perfect nutrient capture efficiency, and in the absence of diffusion, we show that feeding at an angle greatly increases the feeding efficiency of filter feeders. We also show experimental data that living filter feeders in culture feed at an angle to the substrate. We discuss the effects of nutrient diffusion and inefficient nutrient capture on our model, as well as a possible mechanism for filter feeders to change their orientation.

2:05PM CL.00006 The Hungry Fly: Hydrodynamics of feeding in the common house fly¹. MANU PRAKASH, Junior Fellow, Harvard Society of Fellows, MILES STEELE, Deerfield Academy High School — A large number of insect species feed primarily on a fluid diet. To do so, they must overcome the numerous challenges that arise in the design of high-efficiency, miniature pumps. Although the morphology of insect feeding structures has been described for decades, their dynamics remain largely unknown even in the most well studied species (e.g. fruit fly). Here, we use invivo imaging and microsurgery to elucidate the design principles of feeding structures of the common house fly. Using high-resolution X-ray microscopy, we record invivo flow of sucrose solutions through the body over many hours during fly feeding. Borrowing from microsurgery techniques common in neurophysiology, we are able to perturb the pump to a stall position and thus evaluate function under load conditions. Furthermore, fluid viscosity-dependent feedback is observed for optimal pump performance. As the gut of the fly starts to fill up, feedback from the stretch receptors in the cuticle dictates the effective flow rate. Finally, via comparative analysis between the house fly, blow fly, fruit fly and bumble bees, we highlight the common design principles and the role of interfacial phenomena in feeding.

¹We acknowledge financial support from Harvard Milton Fund, Harvard Society of Fellows and Center for Bits and Atoms (MIT).

2:18PM CL.00007 Patterns in Clam Excursion Siphon Velocity According to External Environmental Cues, D.R. WEBSTER, S.K. DELAVAN, Georgia Tech — This study attempts to determine the patterns and/or randomness of the excursion velocity of actively feeding clams, Mercenaria mercenaria. We hypothesize that clams alter their feeding current velocity patterns or randomness according to external cues in the environment such as hydrodynamic characteristics, density of the clam patch, and presence of predators in the upstream flow. A PIV system measured vector fields for two-dimensional planes that bisect the clam excursion siphons, and time records were extracted at the siphon exit position. Fractal and lacunarity analysis of the jet velocity time records revealed that clams alter their jet excursion velocity unsteadiness according to the horizontal crossflow velocity. The results also reveal that the effect of clam patch density on the feeding activity was dependent on the size of the organism. This size/density dependent relationship suggests that predation by blue crabs dominates the system since larger clams are no longer susceptible to blue crab predation, whereas clams of all sizes are susceptible to whelk predation. Finally, clams increase the randomness of their excursion jet velocity when predator cues are located in the upstream flume flow. This suggests that the presence of predators elicits clam behavior that promotes the mixing and dilution of their chemical metabolites.

2:31PM CL.00008 Peristaltic pumping in an elastic tube: feeding the hungry python, DAISUKE TAKAGI, University of Cambridge, NEIL BALMFORTH, University of British Columbia — Biological ducts convey contents like food in the digestive system by peristaltic action, propagating waves of muscular contraction and relaxation. The motion is investigated theoretically by considering a radial force of sinusoidal or Gaussian form moving steadily down a fluid-filled axissymmetric tube. Effects of the prescribed force on the resultant fluid flow and elastic deformation of the tube wall are presented. The flow can induce a rigid object suspended in the fluid to propel in different ways, as demonstrated in numerous examples.

2:44PM CL.00009 Micro-Macro Scale Mixing Interactions by Intestinal Villi Enhance Absorption: a 3D Lattice-Boltzmann Model, YANXING WANG, JAMES BRASSEUR, GINO BANCO, Penn State University — Muscle-induced villi motions may create a micro-scale flow that couples with a lumen-scale macro flow to enhance nutrient transport and absorption in the intestine. Using a 3D multiscale lattice Boltzmann model of a lid-driven cavity flow with microscale 3-D leaf and finger-like villi in pendular motion at the lower surface, we analyze the coupling between micro and macro scale nutrient mixing and absorption at the villi surfaces. RESULTS: The villi motions enhance absorption by creating a micro-mixing layer (MML) that pumps low concentration fluid from between villi groups and attracts fluid with high concentration from the macro flow. The MML couples with the macro flow via a diffusion layer. Leaf-like villi create the strongest MML and, consequently, the highest absorption rates. The finger-like villi create a weaker MML due to the existence of flow between villi. The strength of the MML and nutrient absorption increases with villus frequency. The absorption rate also increases with villus length; however the simulations predict an optimal length close to the physiological length of villi in humans. The complex flow structures that couple micro and macro scale motility-induced fluid motions and macro-scale motility-induced flow may play a significant role in intestinal absorption. Supported by NSF Grant CTS-056215.

2:57PM CL.00010 Contraction-driven Mixing and Absorption in the Intestine with a 3D lattice-Boltzmann Model, BANCO GINO, JAMES BRASSEUR, YANXING WANG, Penn State University — The primary purpose of the intestines is absorption of nutrient molecules and transport of chyme by specific motility patterns. These are broadly classified as segmental (for radial mixing) and peristaltic (for axial transport). Our AIM was to identify an optimal patterning of contractions. METHODS: Fluid motions were modeled in a tubular intestine with a 3D lattice Boltzmann algorithm with prescribed motions of the lumen wall, parameterized from MRI data. Nutrient concentration was modeled as a passive scalar. Scalar was zero at the lumen wall to model rapid nutrient uptake. Simulations were initialized with uniform concentration. RESULTS: The mechanics and more complex than currently believed. Whereas absorption is maximal when fully occluded, there is little differentiation between segmentation and peristalsis at high occlusions. However at lower occlusions absorption is reduced 28% from maximum with segmentation, but as much as 56% with peristalsis. The power requirements are reduced nearly 90% from maximum in segmentation at the lower occlusions! Thus there is a great physiological advantage to choose segmentation at lower occlusion values to enhance mixing and absorption. The MRI measurements indicate that in vivo occlusion values are in the low range where the tradeoff between mechanics and energetics is functionally optimal. Supported by NSF.

Sunday, November 21, 2010 1:00PM - 2:57PM — Session C Microfluids: General II Long Beach Convention Center 202B
1:00PM CM.00001 Computational sensitivity analysis of geometric parameters in laminar superhydrophobic microchannels, ASGHAR YARAHIMADI, MEREDITH METZGER, University of Utah — This talk presents 3-D numerical simulations of laminar flow through a microchannel of height \( h \) containing superhydrophobic surfaces (SHS) along the top and bottom walls. The SHS is modelled as an array of longitudinal shear-aware surfaces having width \( w \) and inclination angle \( \alpha \). The simulations allow for a phase offset \( \ell \) between the shear-aware surfaces respectively on the top and bottom walls. The sensitivity of velocity, wall shear stress, and slip-length with respect to infinitesimal changes in the geometrical design parameters \( (w, \alpha, \ell, \text{ and } h) \) was examined using the Sensitivity Equation Method and Complex Step Differentiation. These techniques differ from traditional parametric studies in that sensitivities are obtained more accurately by direct numerical solution of a separate set of PDEs for the sensitivity derivatives. In this manner, the present sensitivity results can be used to reliably predict the percent drag savings achievable for a unit increase in \( w \) and \( h \). Sensitivity results also indicate that an increase in \( \alpha \) translates into enhanced mixing, albeit with a drag penalty. Finally, the talk discusses how the present sensitivity results may be incorporated into a gradient-based optimization algorithm toward improved microchannel design.

1:13PM CM.00002 Effect of wall pattern configurations on Stokes flow through a microchannel with superhydrophobic slip, H.M. MAK, C.O. NG — The present work aims to study low-Reynolds-number flow through a microchannel with superhydrophobic surfaces, which contain a periodic array of parallel ribs on the upper and lower walls. Mimicking impregnation, the liquid is allowed to penetrate the grooves between the ribs which are filled with an inviscid gas. The array of ribs and grooves gives a heterogeneous wall boundary condition to the channel flow, with partial-slip boundary condition on the solid surface and no-shear boundary condition on the liquid-gas interface. Using the method of eigenfunction expansions and domain decomposition, semi-analytical models are developed for four configurations. Two of them are for longitudinal flow and the others are for transverse flow. For each flow orientation, in-phase and out-phase alignments of ribs between the upper and lower walls are analyzed. The effect of the phase alignments of ribs is appreciable when the channel height is sufficiently small. In-phase alignment gives rise to a larger effective slip length in longitudinal flow. On the contrary, out-phase alignment will yield a larger effective slip length in transverse flow. This work was supported by the Research Grants Council of the Hong Kong Special Administrative Region, China, through Project HKU 7156/09E.

1:26PM CM.00003 Surface Energy Characterization of Superhydrophobic Surfaces under Varying Ambient Temperatures, ARNAV CHHABRA, RAVITEJ KANAPURAM, TAE JIN KIM, CARLOS HIDROVO, University of Texas at Austin — A Cassie state is a state where a liquid droplet sits on top of a rough surface while maintaining a gas layer under the liquid interface. This state decreases the effective solid surface energy, and the contact angle is consequently increased. The contact angle of liquid droplets formed under the Cassie state is a function of the surface roughness and surface energies, dependent primarily on temperature, involved. We studied the surface energy variation of polymeric surfaces under different ambient temperatures, including surfaces that induce Cassie state. A highly rough PDMS (poly-dimethylsiloxane) pillar arrayed surface was used as the substrate. To study these properties, we employed Zisman plots for a number of liquids at different temperatures. A Zisman plot displays variation in contact angles of droplets with different surface energies, thus allowing us to determine the surface energies involved. Such experiments required a methodology that calculated each surface property independently and the subsequent use of image decomposition analyses. We computed the contact angle using similar analyses. These two parameters allowed for the successful construction of the Zisman plot.

1:39PM CM.00004 Streaming potential generated by a pressure-driven flow over a superhydrophobic surface, HUI ZHAO, University of Nevada Las Vegas — The streaming potential generated by a pressure-driven flow over a weakly charged striped slip-stick surface (the zeta potential of the surface is smaller than the thermal potential (25 mV)) with an arbitrary double layer thickness is theoretically studied by solving the Poisson-Boltzmann equation and Stokes equation. A series solution of the streaming potential is derived. Approximate expressions for the streaming potential in the limits of thin double layers and thick double layers are also presented, in excellent agreement with the full solution. The streaming potential is compared against that over a homogeneously charged smooth surface. Our results indicate that the streaming potential over a superhydrophobic surface only can be enhanced when the liquid-gas interface is charged. In addition, as the double layer thickness increases, the advantage of the super-hydrophobic surface diminishes. The impact of a slip-stick surface on the streaming potential might provide guidance for designing novel and efficient microfluidic energy conversion devices using a super-hydrophobic surface.

1:52PM CM.00005 Modeling thermocapillary flow on superhydrophobic surfaces, TOBIAS BAIER, CLARISSA STEFFES, STEFFEN HARDT, Technische Universität Darmstadt — A liquid in Cassie-Baxter state confined between two microstructured superhydrophobic surfaces has a large free surface fraction while at the same time being in close contact to a solid. As such this configuration is predestined for thermocapillary transport when applying a temperature gradient along the structured substrate. We analytically and numerically investigate the thermocapillary flow over superhydrophobic arrays of fins, with the temperature gradient applied in parallel and transverse directions. At the gas-solid interface Marangoni stresses exert a force on the liquid driving it towards the regions of higher surface tension. This leads to a spatially varying near-wall flow profile extending a distance of the order of the fin spacing into the liquid, while a plug flow prevails in the bulk of the liquid. In the Stokes limit we are able to relate the bulk flow velocity to the hydrodynamic slip length for pressure-driven flow over such surfaces. The numerical results indicate that this relation serves as an upper bound for the achievable flow velocities at temperature gradients. Since even moderate temperature gradients of the order of a few K/cm can induce flow velocities of several mm/s for water-based systems, this setup lends itself for microfluidic pumping.

2:05PM CM.00006 3-D CFD Simulations of Liquid Laminar Flow over Superhydrophobic Surfaces for two scenarios: (1) Shear-Driven Flow with Square Post Geometries, (2) Pressure-driven Flow with Rectangular Post Geometries, ABOLFALZI AMIN, DAN MAYNES, BRENT WEBB, Brigham Young University — We numerically investigate the influence of post patterned superhydrophobic surfaces on the drag reduction for liquid flow through microchannels. Hydrophobically coated surfaces exhibiting microscale structures such as ribs/cavities and posts/cavities can significantly reduce the liquid-solid contact. Preventing liquid from entering the cavities increases the fraction of liquid-gas interfaces, which reduces friction between liquids and solid walls. This results in reduced surface friction. Fully developed steady laminar flow for the two scenarios is considered here. The effects of aspect ratio, cavity fraction, and relative module width on the slip length and on the Darcy friction factor-Reynolds number product, \( \text{Re} \), were explored numerically. Various aspect ratios, cavity fractions, and relative module widths were explored. The present results are compared with those for surfaces exhibiting square posts in pressure-driven liquid laminar flow. As the aspect ratio of the posts increased or decreased, the \( \text{Re} \) values asymptotically approached those of surfaces exhibiting longitudinal and transverse ribs, respectively.

2:18PM CM.00007 Amplification of the electroosmotic velocity by induced charges at fluidic interfaces, CLARISSA STEFFES, TOBIAS BAIER, STEFFEN HARDT, Center of Smart Interfaces, Technische Universität Darmstadt, Germany — The performance of microfluidic devices like electroosmotic pumps is strongly limited by drag forces at the channel walls. In order to replace the standard no-slip condition at the wall with a more favorable slip condition, superhydrophobic surfaces are employed. In the Cassie-Baxter state, air is entrapped in the surface cavities, so that a significant fraction of water-air interfaces at which slip does occur is provided. However, such surfaces do not enhance electroosmotic flow. Since no net charge accumulates at the liquid-air interfaces, the driving force is reduced, and no flow enhancement is obtained. We consider electrodes incorporated in the superhydrophobic structure to induce charges at these interfaces, thereby increasing the driving force. A theoretical model is set up, yielding an understanding of the influence of the surface morphology on the flow, which serves as a basis for ongoing experimental work. While a considerable enhancement of the electroosmotic velocity is already expected for standard superhydrophobic surfaces, greater amplifications of one order of magnitude may be achieved by substituting the air in the surface cavities by oil, reducing the risk for electric breakdown or transition to the unfavorable Wenzel state.
2:31PM CM.00008 Dynamics of the Liquid Meniscus in Micropillar Arrays, RONG XIAO, RYAN ENRIGHT, EVELYN WANG, MIT — Liquid dynamics in superhydrophilic micropillar arrays is of broad interest in microfluidics for lab-on-a-chip, biomedical, and thermal management applications. Accurate prediction and optimization of propagation rates in such microstructures require detailed understanding of the evolution of the liquid meniscus. In this work, we experimentally investigated microfabricated circular pillar arrays with diameters of 2.5 μm and 5 μm, periods ranging from 5 μm to 30 μm, and heights ranging from 10 μm to 30 μm. By coupling interference microscopy and high-speed imaging, the dynamics of the advancing liquid front were precisely captured. Two distinct time scales in the wetting process were observed associated with the liquid sweeping across the bottom surface and rising along the sides of the pillars, which is dependent on the height-to-period ratio of the pillar array. This behavior was modeled by using an energy-based approach. This work provides important insights towards accurately predicting propagation rates for a range of micropillar arrays and can be extended to other microstructure geometries.

2:44PM CM.00009 Disjoining pressure in thin liquid films on charged structured surfaces, CHRISTIAAN KETELAAR, VLADIMIR AJAEV, Southern Methodist University — We consider thin liquid films on various structured surfaces and compute the electrostatic component of disjoining pressure in the film. The regions of solid phase in contact with the liquid are assumed to be at a constant electrical potential. Presence of ions in the liquid implies that the electrical field there is described by the Poisson-Boltzmann equation. Situations are considered when liquid fills the spaces between the elements of the structure (e.g. grooves) and when pockets of air remain trapped there. The formulas for disjoining pressure are incorporated into a numerical method for calculation of deformations of air-liquid interfaces. Applications of our mathematical model to recent experiments on evaporation of thin liquid droplets on structured surfaces are discussed.

Sunday, November 21, 2010 1:00PM - 3:10PM — Session CN Vortex Dynamics and Vortex Flows II Long Beach Convention Center 202C

1:00PM CN.00001 Mechanics of Viscous Vortex Reconnection, F. HUSSAIN, U. of Houston, K. DURAIASAMY, Stanford U. — This work builds on our long-standing claim that reconnection of coherent structures is the dominant mechanism of jet noise generation and that reconnection plays a key role in both energy cascade and fine-scale mixing in fluid turbulence. Reconnection of two anti-parallel vortex tubes is studied by direct numerical simulations of the incompressible Navier-Stokes equations over a wide range (250-9000) of the vortex Reynolds number (Re). Reconnection is never complete, leaving behind a part of the initial tubes as “threads,” which then undergo successive reconnections (our cascade and mixing scenarios) as the newly formed “bridges” recoil from each other by self-advection. We find that the time tR for orthogonal transfer of circulation scales as tR ≈ Re−3/4. The shortest distance d between the tube centroids scales as d ≈ a(Re(t0 − t))1/3, before reconnection and as d ≈ b(Re(t − t0))2 after reconnection. Bridge repulsion is faster than collision and has less variation as local induction predominates, and is clearly the most intense sound generation phase. The maximum rate of vortex circulation transfer, enstrophy production and dissipation scale as Re3, Re1/4, Re−1/2, respectively.

1:13PM CN.00002 Regenerative transient growth on a vortex column1, ERIC STOUT, FAZLE HUSSAIN, U. Houston — Perturbations on a Lamb-Oseen vortex column with a circulation overshoot (due to a sheath of negative axial vorticity. −ωz, surrounding the core) are studied by DNS of the incompressible Navier-Stokes equations, for a range (500-12500) of the vortex Reynolds number (Re= circulation/viscosity). Initial perturbation radial (rad.) vorticity is tilted by the mean strain into perturbation azimuthal (az.) vorticity, which generates positive Reynolds stress necessary for energy growth. The meridional flow of az. vorticity tilts −ωz into intensifying rad. vorticity, increasing the +Reynolds stress, which results in exponential energy growth. +Reynolds stress also transports azimuthal momentum radially outward, reducing the overshoot magnitude, which determines −ωz. The resulting decreased rad. vorticity reduces the +Reynolds stress, arresting instability growth (with concomitant increase in viscous dissipation). Outward transport of azimuthal momentum also produces +Reynolds stress, which then transports azimuthal momentum inward. A new circulation peak appears nearer to the column axis, initiating a period of new, regenerative transient growth — a promising scenario for turbulence generation near the vortex column.

1:26PM CN.00003 ABSTRACT WITHDRAWN —

1:39PM CN.00004 A short wave instability caused by the approach of a vortex pair to a ground plane, DANIEL M. HARRIS, Massachusetts Institute of Technology, CHARLES H.K. WILLIAMSON, Cornell University — In the present work, we experimentally study the approach of a counter-rotating vortex pair to a ground plane. The trajectories of the primary vortices in experiment differ quite significantly from the inviscid case, primarily due to the fact that between the vortices and the ground, a boundary layer forms, which can separate to generate jets. Experiments on the approach of a counter-rotating vortex pair to a ground plane were observed associated with the liquid sweeping across the bottom surface and rising along the sides of the pillars, which is dependent on the height-to-period ratio of the pillar array. This behavior was modeled by using an energy-based approach. This work provides important insights towards accurately predicting propagation rates for a range of micropillar arrays and can be extended to other microstructure geometries.

1:52PM CN.00005 A unified criterion for the centrifugal instability of vortices and swirling jets, PAUL BILLANT, LadHyX, CNRS-Ecole Polytechnique, F-91128 Palaiseau, France, FRANCOIS GALLAIRE, LFMI, EPFL, 1015 Lausanne, Switzerland — It is well known that swirling jets can become centrifugally unstable like pure vortices but with a different azimuthal wavenumber selection. The Leibovich and Stewartson (1983) criterion is a generalization of the Rayleigh criterion to swirling jets: it is a sufficient condition for instability with respect to perturbations with both large axial and azimuthal wavenumbers. We have relaxed the large azimuthal wavenumber assumption in this criterion and obtained a new criterion that is valid whatever the azimuthal wavenumber and whatever the magnitude of the axial flow: from zero (pure vortex) to finite values (swirling jets). The new criterion recovers the Leibovich-Stewartson criterion when the azimuthal wavenumber is large and the Rayleigh criterion when the azimuthal wavenumber is small. The criterion is confirmed by comparisons with numerical stability analyses of various classes of swirling jet profiles. In the case of the Batchelor vortex, it provides more accurate results for perturbations with finite azimuthal wavenumbers than the Leibovich-Stewartson criterion. The criterion shows also that a whole range of azimuthal wavenumbers are destabilized as soon as a non-zero axial velocity component is present in a centrifugally unstable vortex.
2:05PM CN.00006 Secondary Flows Within 3D Vortices. SUYANG PEI, PEDRAM HASSANZADEH, PHILIP MARCUS, University of CA at Berkeley — Control volume analyses, analytic scaling arguments, and numerical modeling can all be used to show that in a dissipationless flow that there are classes of 3D vortices in which the fluid velocity is purely 2D. That is, in cylindrical coordinates the vortex occupies a finite region in \( \rho \) and \( \phi \) directions. However, control volume analyses and scaling arguments show that if dissipation is present, these 3D vortices must have a 3D meridional, or secondary, circulation. The dissipation can be due to viscosity, or if the flow is temperature- or salt-stratified, the dissipation can be due to the diffusion of heat or salt. The best known secondary circulation in a 3D vortex is Ekman pumping, which requires that the 3D vortex is confined between upper and lower solid boundaries. We present numerical results of secondary flows within several classes of 3D vortices, including not bound vortices that are not bound above and below by solid walls. We relate these results to geophysical vortices, including ocean eddies and planetary vortices.

2:18PM CN.00007 How Do 3D Vortices Spin Down, or Do They? . PEDRAM HASSANZADEH, SUYANG PEI, PHILIP MARCUS, University of CA at Berkeley — It is well known that laminar 3D vortices sandwiched between upper and lower boundaries in a rapidly rotating flow will rapidly spin down due to Ekman pumping. The pumping transports angular momentum and energy out of the vortex and into thin boundary layers where viscosity acts efficiently. On the other hand the Great Red Spot of Jupiter and laboratory models of the Red Spot, which are 3D vortices sandwiched between upper and lower boundaries in a rapidly rotating tank, do not spin down. Those 3D vortices maintain themselves indefinitely. The longevity of the Red Spot has been attributed to angular momentum transfer from its surrounding shearing flow, from its mergers with smaller vortices, and to a number of other processes, none of which have been verified and all of which lack plausibility. The reasons behind the longevity of “Red Spots” in the laboratory have never been examined. We present numerical results that show how a laboratory model of the Red Spot maintains itself and does not spin down.

2:31PM CN.00008 Control of vortex breakdown in a contracting pipe . FRANCOIS GALLAIRE, PHILIPPE MELICA, Ecole Polytechnique Federale de Lausanne- Lab. of Fluid Mechanics and Instabilities — We investigate numerically the vortex breakdown of a viscous, swirling jet developing in an axisymmetric contracting pipe. When the swirl number, i.e. the ratio of the maximum azimuthal to streamwise velocity, exceeds a certain threshold value, such flows are known to undergo a violent transition from the so-called columnar state to the breakdown state, the latter being characterized by a large recirculation region. We show first that breakdown occurs through a saddle-node bifurcation, owing to the destabilization of an axisymmetric periodic solution. This scenario also holds when the wall has a small curvature, which is also allowed in our model. Such a control technique can be easily implemented in practice and is originally designed to restore the linear stability of the bifurcating mode close to threshold. Results issuing from fully nonlinear simulations will be presented. In particular, it will be shown that vortex breakdown can be suppressed over a large range of swirl numbers, even for low-flow-rate jets representing only a few per cent of the flow rate in the inlet section.

2:44PM CN.00009 Structure of a Steady Bathtub Vortex. ANDERS ANDERSEN, Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, DK-2800 Kgs. Lyngby, Denmark, Lasse Bøhling, Glass and Time, IMFUFA, Roskilde University, DK-4000 Roskilde, Denmark, DAVID FABRE, Universite de Toulouse, INPT, UPS; Institut de Mecanique des Fluides de Toulouse (IMFT), Allee du Professeur Camille Soulia, F-31400, Toulouse, France — Bathtub vortex flows constitute an important class of concentrated vortex flows which are characterised by intense axial down-flow and stress free surface. We use direct numerical simulations to explore the flow structure of a steady bathtub vortex in a cylindrical tank with a central drain-hole. We find that the qualitative structure of the meridional flow does not depend on the radial Reynolds number, whereas we observe a weak overall rotation at low radial Reynolds number and a concentrated vortex above the drain-hole at high radial Reynolds number. We present a simple analytical model which shows the same qualitative dependence on the radial Reynolds number as the simulations and which compares favourably with the results for the radial velocity and the azimuthal velocity at the surface. Finally, we describe the height dependence of the radius of the vortex core and the maximum of the azimuthal velocity at high radial Reynolds number, and we show that the data on the radius of the vortex core and the maximum of the azimuthal velocity as functions of height collapse on single curves by appropriate scaling.

2:57PM CN.00010 Vortex Behavior in Fully-Oscillating Low-Speed Jet Flows1. PRESTON JONES, JOHN BAKER, University of Alabama — Vortex formation associated with a fully oscillating low-speed jet was studied to better understand the fundamental nature of such flows. It has been hypothesized that vortices produced by sinusoidal flow from a nozzle will behave in a manner different from that observed for typical piston-cylinder generated vortices. A variable speed reciprocating pump, designed to produce sinusoidal flow fields at the nozzle exit, was used to examine vortex characteristics as a function of Reynolds number and dynamic vortex formation number. The behavior was visualized using a passive scalar dye. Video recording were used to examine the nature of the flows for the above-mentioned dimensionless parameters. Flows corresponding to Reynolds numbers in the range of 244 to 2708 and dynamic vortex formation numbers in the range of 0.82 to 62.92 were considered. The fully oscillating jets flows produced vortices that appear to not exhibit the critical vortex formation number of 4, commonly observed for pulsating jets. Reynolds number was shown to have an impact on physical vortex detachment.

1Work performed under REU site sponsored by NSF grant EEC 0754117.

Sunday, November 21, 2010 1:00PM - 3:10PM – Session CP Nanofluids II Long Beach Convention Center 203A

1:00PM CP.00001 Pump-Free Composite Nanochannels for Chip-Level Cooling1. ZHIGANG LI, CHONG LIU, Department of Mechanical Engineering, The Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong — In this work, we propose a composite nanochannel system, where half of the channel is of low surface energy, while the other half has relatively high surface energy. It is shown that fluids in such channels can be continuously driven by a symmetric temperature gradient. In the low surface energy part, the fluid moves from high to low temperature, while the fluid migrates from low to high temperature in the part of high surface energy. The mechanisms that govern the flow are explained and the conditions require. To guarantee the flow and the possible applications are discussed. One advantage about this system is the application for chip-level cooling, where the heat generated in the chip can be used to drive the liquid without using external pumps, which consume energy, occupy space, and therefore conflict with the miniaturization objectives of the next generation electronic devices.

1Direct Allocation Grant No. DAG07/08.EG02 and School Initiative Grant No. SB107/08.EG01.

1:13PM CP.00002 Field amplified sample stacking and focusing in nanofluidic channels . BRIAN STOREY, Olin College, JESS SUSTARIICH, SUMITA PENNATHUR, UC Santa Barbara — One major obstacle in the widespread adoption of nanofluidic technology for bioanalytical systems is efficient detection of samples due to the inherently low numbers of molecules present in small channels. This work explores one of the most common preconcentration techniques, field-amplified sample stacking (FASS), in nanofluidic systems in efforts to alleviate this obstacle. Holding the ratio of background electrolyte concentrations constant, the parameters of channel height, strength of electric field, and electrolyte concentration are experimentally varied. Although in micro scale systems these parameters have little or no effect on the final concentration enhancement achieved, nanofluidic experiments show strong dependencies on each of these parameters. Further, nanofluidic systems demonstrate an increased concentration enhancement over what is predicted and realized in micro-scale counterparts. Accordingly, a theoretical model is developed that explains these observations and furthermore predicts a novel focusing mechanism that can explain the observed increase in concentration enhancement. The simple model is capable of predicting key experimental observations, while a model that incorporates more detail provides good comparisons to the experiment.
1:26PM CP.00003 ABSTRACT WITHDRAWN –

1:39PM CP.00004 Investigation of flow velocity profile in a nanocapillary ¹, GUIREN WANG, AN ZOU, FANG YANG, University of South Carolina, Columbia — In order to understand transport phenomena in nanofluidics, we study the flow velocity profile in a nanocapillary. Laser Induced Fluorescence Photobleaching Anemometer (LIFPA) and Stimulated Emission Depletion (STED) are combined to establish a far-field nanoscopic velocimeter for flow velocity measurement in a nanochannel. LIFPA uses molecular dye as tracer to avoid issues involved in particles as tracer in PIV, when one dimension of the channels is nearly in the same order of particle diameter. STED is applied to overcome conventional diffraction limit in physics to increase spatial resolution. To apply LIFPA, calibration is first required to establish the relationship between the fluorescence intensity signal and flow velocity. Current monitoring, time-fly of fluorescence and metering from syringe pump are used for the calibration respectively. Then the velocity profile is measured with a spatial resolution of about 70 nm in a nanocapillary with inner diameter of 360 nm. The conductivity influence on the velocity profile is investigated.

1:52PM CP.00005 Temperature and viscosity effects on the velocity profile of a nanochannel electro-osmotic flow¹, BOHUMIR JELINEK, SERGIO D. FELICELLI, Mississippi State University, PAUL F. MLAKAR, JOHN F. PETERS, ERDC, Vicksburg MS — Significant temperature and viscosity effects on the electrokinetic transport in a nanochannel with a slab geometry are demonstrated using a molecular dynamics (MD) model. A previously studied system consisting of Na⁺ and Cl⁻ ions dissolved in water and confined between fixed crystalline silicon walls with negatively charged inner surfaces in an external electric field was investigated. Lennard-Jones (LJ) force fields and Coulomb electrostatic interactions with Simple Point Charge Extended (SPC/E) model were used to represent the interactions between ions, water molecules, and channel wall atoms. Dependence of the flow of water and ions on the temperature was examined. The magnitude of the water flux and even its direction are shown to be significantly affected by temperature. Temperature dependence of the flux was attributed to the charge redistribution and to the changes in viscosity of water. Using a simple inverse power approximation for water viscosity profile across the channel instead of constant viscosity, an improved prediction of MD electro-osmotic velocity profile from charge density by Stokes equation is demonstrated.

²Supported by NSF CAREER CBET 0644719

2:05PM CP.00006 Hydrodynamic rotational friction of single-wall carbon nanotubes in liquid suspension¹, JERRY SHAN, FRANK ZIMMERMANN, Rutgers University — The hydrodynamic resistance to rotation in liquid suspension of single-wall carbon nanotubes 1 nm in diameter was experimentally investigated and compared with theoretical predictions. Nanotubes were forced to rotate into alignment with an external electric field, and the rate of their alignment response was measured with laser polarimetry. The measured rates of change of the nematic-order parameter were approximately consistent with theoretical predictions based on classical, no-slip hydrodynamics. This implies that, despite the reduced resistance previously reported for internal flow through carbon nanotubes, and the fact that the nanotubes’ diameter is of the same order as the size of the solvent molecules, classical continuum hydrodynamics essentially holds for external flow about individual single-wall carbon nanotubes in liquids.

2:18PM CP.00007 Nanofluid heat transfer enhancement in a developing laminar shear flow. J.T.C. LIU, School of Engineering, Brown University — The continuum conservation equations for nanofluid flow (J. Buongiorno 2006 J. Heat Transfer 128, 240-50) is applied to a two-dimensional channel entrance region, subjected to a Rayleigh approximation for the nonlinear advection. A perturbation expansion for very small nanoparticle volume concentration is used to further simplify the system. The zeroth order similarity solution furnishes the input for the first order problem for nanofluid momentum, volume concentration and heat transport. The latter is cast into a form to show the effect of volume concentration as an effective heat source, thus promoting enhanced heat transfer. Similar solutions for the sequentially solvable first order system is sought to quantitatively describe the dynamics and thermodynamics of nanofluid flow in this much simplified shear flow problem.

2:31PM CP.00008 ABSTRACT WITHDRAWN –

2:44PM CP.00009 Evanescent wave based near-wall thermometry utilizing Brownian motion¹, KANJIRAKAT ANOOP, RANA KHADER, REZA SADR, Texas A&M at Qatar — Near wall velocity and temperature measurement is instrumental in research associated with convection heat transfer. Nano Particle Image Velocimetry (nPIV) technique is known to be an effective tool for near-wall velocity measurements. nPIV uses evanescent wave generated by total internal reflection of light to illuminate particles within few hundred nanometers of the wall. Furthermore, temperature measurement at micro scale using Brownian motion characteristics of sub-micron tracer particles used in Micro PIV is well established. This temperature measurement technique is based on the fact that a change in temperature affects Brownian motion that consequently affects the PIV cross-correlation characteristics. In this study the possibility of utilizing this effect for near-wall thermometry is investigated using synthetic nPIV images of 100nm diameter particles. In addition to Hindered Brownian motion, the numerical method includes other near wall forces on the particles such as shear induced lift, buoyancy, electrostatic repulsion, and van der Waals attraction. Simple experiments are carried out using stationary liquids at different temperatures to verify the obtained results.

³Supported by QNRF.

2:57PM CP.00010 Hybrid Continuum and Molecular Modeling of Nano-scale Flows, ALEX POVITSKY, SHUNLIU ZHAO, Carleton University — A novel hybrid method combining the continuum approach based on boundary singularity method (BSM) and the molecular approach based on the direct simulation Monte Carlo (DSMC) is developed and then used to study viscous fibrous filtration flows in the transition flow regime, Kn > 0.25. The DSMC is applied to a Knudsen layer enclosing the fiber and the BSM is employed to the entire flow domain. The parameters used in the DSMC and the coupling procedure, such as the number of simulated particles, the cell size and the size of the coupling zone are determined. Results are compared to the experiments measuring pressure drop and flowfield in filters. The optimal location of singularities outside of flow domain was determined and results are compared to those obtained by regularized Stokeslets. The developed hybrid method is parallelized by using MPI and extended to multi-fiber filtration flows. The multi-fiber filter flows considered are in the partial-slip and transition regimes. For Kn~1, the computed velocity near fibers changes significantly that confirms the need of molecular methods in evaluation of the flow slip in transitional regime.

Sunday, November 21, 2010 1:00PM - 3:10PM –
Session CQ Biolocomotion I: Micro-swimming I Long Beach Convention Center 203B
1:00PM CQ.00001 Diffusion vs. locomotion. ERIC LAUGA, UCSD — In this talk we consider small organisms self-propelling in viscous fluids. We address theoretically and numerically the interplay between fluid-based locomotion and Brownian motion. Interesting dynamics occurs on time scales close to, or larger than, the inverse rotational diffusion constant for the organism, where the cells transition from swimming to diffusing. We derive results valid for all types of swimmers, including a new diffusion constant.

1:13PM CQ.00002 Stirring by squirmers. JEAN-LUC THIFFEaulT, University of Wisconsin, ZHI LIN, IMA, University of Minnesota, STEPHEN CHILeddRESS, Courant Institute, NYU — We analyze a simple “Stokesian squirmer” model for the enhanced mixing due to swimming micro-organisms [1]. The model is based on a calculation of Thiffeault & Childress [2], where fluid particle displacements due to inviscid swimmers are added to produce an effective diffusivity. Here we show that for the viscous case the swimmers cannot be assumed to swim an infinite distance, even though their total mass displacement is finite. Instead, the largest contributions to particle displacement, and hence to mixing, arise from random changes of direction of swimming and are dominated by the far-field stresslet term in our simple model. We validate the results by numerical simulation. We also calculate nonzero Reynolds number corrections to the effective diffusivity. Finally, we show that displacements due to randomly-swimming squirmers exhibit PDFs with exponential tails and a short-time superdiffusive regime, as found previously [3]. In our case, the exponential tails are due to “sticking” near the stagnation points on the squirmer’s surface.

1:26PM CQ.00003 Hydrodynamic interaction of two unsteady squirmers. TAKUJI ISHIKAWA, DAVIDE GIACCE, Tohoku University — The study of pair-wise interactions between swimming microorganisms is fundamental to the understanding of the rheological and transport properties of semi-dilute suspensions. In this study, the hydrodynamic interaction of two ciliated microorganisms is investigated numerically using a boundary-element method. The microorganisms are modeled as spherical squirmers that swim by time-dependent surface deformations. The results show that the inclusion of the unsteady terms in the ciliary propulsion model has a large impact on the trajectories of the interacting cells, and causes a significant change in scattering angles with potential important consequences on the diffusion properties of semi-dilute suspensions. Furthermore, the analysis of the shear stress acting on the surface of the microorganisms revealed that the duration and the intensity of the near-field interaction are significantly modified by the presence of unsteadiness. This observation may account for the hydrodynamic nature of randomness in some biological reactions, and surpases the distinction between intrinsic randomness and hydrodynamic interactions, adding a further element to the understanding and modeling of interacting microorganisms.

1:39PM CQ.00004 Breaking Symmetry with Gravity: Two-Link Swimming Using Buoyant Orientation. LISA BURTON, Massachusetts Institute of Technology, ROSS HATTON, HOWIE CHOSER, Carnegie Mellon University, ANETTE HOSOI, Massachusetts Institute of Technology — Swimming at low Reynolds number requires the swimmer’s motion to be non-reciprocal in order to break the time-reversal symmetry of the equations of motion. We demonstrate that a neutrally buoyant swimmer can achieve net motion simply by introducing a static separation between the centers of mass and buoyancy. In the presence of gravity, the swimmer passively reorients towards its natural equilibrium without changing shape. We derive the governing equations for the system and explain how to control swimming direction with parameter and stroke selection and discuss swimming efficiency for various strokes.

1:52PM CQ.00005 Low-Reynolds-number swimming in confined geometries. DARREn CROWDY, Imperial College London — We present results of a theoretical investigation into the locomotion in confined geometries (e.g. near no-slip walls) of simple circular swimmers, in two dimensions, actuated by some imposed velocity profile in their surface. It is shown how use of the reciprocal theorem of Stokes flows together with knowledge of exact solutions for certain “dragging problems” can lead to the derivation of explicit dynamical systems for the swimmer’s linear and angular velocities.

2:05PM CQ.00006 Analytical model of a butterfly micro-swimmer. MAKOTO IIMA, ALEXANDER MIKHAIlov — We propose a simple mechanical model consisting of two spheroids (wings) connected by a single hinge. Unlike micro-swimmers proposed so far, this model has just one hinge, but its motion allows two degree of freedom, corresponding to open-close and twisting. Its non-reciprocal operation cycles resembles conformational motions characteristic for real protein machines and similar to the propulsion pattern of a butterfly. The net velocity and the net stall force are calculated analytically and their dependence on the model parameters is discussed.

2:18PM CQ.00007 Acceleration of swimming bacteria at “zero” Reynolds number. JOHN KESSLER, LUIS CISNEROS, University of Arizona, SUJOY GANGULY, RAYMOND GOLDSTEIN, University of Cambridge — Self-propelled objects can accelerate at “zero” Reynolds number Re. An incompressible fluid responds “instantaneously” and globally to the motion of bounding surfaces. When the propulsion mechanism of a bacterial body, a helical bundle of flagella, forms and rotates, the body accelerates according to $F = ma$, where $F$ is the sum of forces exerted on the body; drag plus the thrust of the flagella. The flagellar rotation instantly moves all the surrounding fluid, as does the body's motion on, acting on its surarround. The acceleration of bacteria stopped by a collision and beginning reverse swimming is important in the analysis of the jammed phase in the onset of Zooming BioNematic (ZBN). An instantaneous one step displacement has been used to analyze the flow surrounding swimming bacteria.

2:31PM CQ.00008 Teaching Stokesian Dynamics to Swim. JAMES SWAN, JOHN BRADY, California Institute of Technology — We develop a generic framework for modeling the hydrodynamic self-propulsion (i.e. swimming) of bodies (e.g. microorganisms) at low Reynolds number via Stokesian Dynamics simulations. In this framework, the swimming body is composed of many spherical particles constrained to form an assembly. We map the resistance tensor describing the hydrodynamic interactions among the particles onto that for the assembly. Specifying a particular swimming gate number via Stokesian Dynamics simulations. In this framework, the swimming body is composed of many spherical particles constrained to form an assembly.
2:44PM CQ.00009 Numerical simulations of microscale steady streaming using viscous vortex particle method1, KWITAE CHONG, JEFF D. ELDREDGE, Mechanical & Aerospace Engineering, University of California, Los Angeles, CA, USA — Microscale steady streaming created by localized cyslic boundary deformation provides an appealing option in microfluidic systems. High-frequency oscillatory body motion creates a large-scale circulatory motion in viscous fluid that is ‘steady’ compared to the timescale of oscillation, and this overall net flow can be used for manipulating discrete objects in a micro system. A typical steady streaming motion generated by one or more unidirectionally oscillating cylindrical probes is considered. A high-fidelity numerical approach is presented for simulating such problems using a viscous vortex particle method. By focusing on vorticity, which is confined to a narrow stokes layer surrounding each probe, the method gains computational efficiency over a typical grid-based method. In particular, the large-scale streaming motion can be computed as a post-processing step, and little additional effort is required for multiple probes. Parametric studies of varying geometric arrangement were conducted and reveal the microscale flow structures and particle transport.

2:57PM CQ.00010 Flow shear induced cross-stream migration by a green alga1, ANWAR CHENGALA, Student, MIKI HONDZO, JIAN SHENG, Professor — Swimming and migration characteristics of micro-organisms in shear flows has overarching implications in formation of biological thin layers in aquatic ecosystems, design of bioreactors, and cell separations. Experiments are conducted in a microfluidic channel using digital holographic microscopy. A motile micro-alga, Dunaliella primolecta, is studied in a laminar shear flow at maximum shear rates ranging from 0.1 to 25 s−1. It is found that D. primolecta cells aggregate in the direction of positive vorticity when a critical local shear rate of 5 s−1 is reached. Unlike nonmotile cells, D. primolecta in high shear flow do not rotate along the Jeffrey orbits, neither resumes the local vorticity of flow. The torque on cell body is counter-acted by the spatial alignment of beating flagella. It is speculated that under severe viscous stresses, motile cells “opt” to align themselves in the direction where the least stresses are experienced on cell wall. Beating of flagella, which prevents cells from assuming local flow vorticity, consequently propel them in the span wise direction and allow them to disperse only in a thin two-dimensional layer.

Sunday, November 21, 2010 1:00PM - 2:57PM — Session CR Drops II Long Beach Convention Center 203C

1:00PM CR.00001 Effect of Viscosity and Size of a Droplet on Spreading Dynamics in Electrowetting1, JIWOO HONG, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — Electrowetting-based devices require fast, stable and accurate positioning of the three-phase contact line (TCL). To meet this requirement, a concrete understanding on dynamics of electrowetting is necessary, which has been one of the main challenges in electrowetting research. In this work, we investigated the switching dynamics of a droplet in air actuated by electrowetting, for different applied voltages, drop sizes, and viscosities. We analyzed the spreading process for weakly viscous droplet and derived a relationship between the drop size and the switching time. The relationship is verified experimentally. We explored experimentally the effect of viscosity of a droplet on switching dynamics. During the initial spreading period, the spreading dynamics was hardly dependent on viscosity. The switching time is moderately dependent on fluid viscosity.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) (Grant No. R0A-2007-000-20098-0).

1:13PM CR.00002 Droplet jumping by resonant AC electrowetting envisioning three-dimensional digital microfluidics1, SEUNG JUN LEE, SANGHYUN LEE, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — We introduce droplet jumping by resonant AC electrowetting (DJ-RACE) to transport droplets to vertical direction, envisioning three-dimensional digital microfluidics. Inphase oscillatory actuation by resonant AC electrowetting allows droplets to store sufficient energy for jumping on their stretched surfaces by conventional electrowetting methods. The detailed jumping mechanism is explained in comparison to experimental results, and the actual droplet transport from the superhydrophobic bottom surface to higher level surfaces is demonstrated by several electrode configurations and actuation methods.

1This work was supported by the National Research Foundation of Korea (NRF) grant No.20090083510 funded by the Korean government (MEST) through Multi-phenomena CFD Engineering Research Center.

1:26PM CR.00003 Jumping number in the droplet jumping by resonant AC electrowetting1, SANGHYUN LEE, SEUNG JUN LEE, KWANG HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — The droplet jumping by resonant AC electrowetting (DJ-RACE) is recently introduced to transport droplets to vertical direction, whereby three-dimensional digital microfluidics are envisioned. In DJ-RACE, the central mechanism of the droplet jumping is the conversion of the surface energy stored by resonant AC electrowetting to the kinetic energy for jumping. Here, we newly introduce the jumping number \( J_u=\gamma/\rho g R^2 \), measuring the energy conversion in the jumping process and, thus, the feasibility of droplet jumping. \( J_u \) interprets that droplets having higher \( J_u \) can make higher and easier jumping, and smaller and lighter droplets with higher surface tension can have higher \( J_u \). Practically, \( J_u \) should be greater than 1.5 for the droplet jumping, and active jumping was observed when \( J_u \) is greater than 5. In addition, \( J_u \) can predict the effect of diverse physicochemical changes in a system such as enzymatic additives or impurities on jumping, where it can also provide diverse strategies to compensate these changes. The newly introduced \( J_u \) could be the fundamental and useful parameter in the three-dimensional digital microfluidic devices based on DJ-RACE.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) (Grant No. R0A-2007-000-20098-0).
2Corresponding author

1:39PM CR.00004 Electrophoresis of a charged electrolyte droplet1, DO JIN IM, JIHOON NOH, IN SEOK KANG, POSTECH — We study the motion of a charged droplet in a dielectric fluid under electric field for the use as a microdroplet actuation method. The amount of electrical charging has been measured experimentally for a water droplet and an electrolyte droplet using the Stokes law. Comparison is made with theoretical value of a perfect conductor sphere. The effects of droplet size, electric field strength, and electrolyte concentration on the motion of aqueous droplets are investigated. A scaling law derived from experimental results shows that the amount of charging of de-ionized water droplet is proportional to the square of droplet radius and to electric field strength. This means de-ionized water droplet follows well the scaling law of perfect conductor. However, the electric charging characteristic of electrolyte droplet depends on electrolyte concentration and electric field strength. But, under sufficiently high electric field, electrolyte droplet behaves like perfect conductor regardless of its concentration. This fact implies that the same actuation speed can be obtained independent of electrolyte concentration under high electric field. This property may be utilized for stable actuation of electrolyte droplets.

1This work was supported by the grant R01-2009-0083830 from National Research Foundation (NRF) of Korea, and by the BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.
of angle of nanocrystal superlattice domains orientation.

of vortices quickly decreases with time, resulting in three bulk vortices in the intermediate stage. The vortex structure finally evolves into the single convection near-surface vortices in the drop is controlled by the Marangoni cell size, which is calculated similar to that given by Pearson for flat fluid layers. The number array of vortices arises near a surface of the drop and induces a non-monotonic spatial distribution of the temperature over the drop surface. The number of sessile drop are obtained. We jointly take into account the hydrodynamics of an evaporating sessile drop, effects of the thermal conduction in the drop and the LEV SHCHUR, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory.

On the other hand, where the local direction of electric field is tangential to the drops surface (equator), they form chains that are aligned parallel to the electric dielectrophoretic force the particles also interact with each other via the dipole-dipole interactions to form chains or move away from each other depending on This technique could be useful to concentrate particles at a drop surface, and also separate two types of particles. In this talk we show that in addition to the dielectrophoretic force that acts upon particles because the electric field on the surface of the drop is non-uniform, despite the fact that the applied electric field is uniform.

This motion is due to the dielectrophoretic force which acts upon particles because the electric field on the surface of the drop is non-uniform, despite the fact that the applied electric field is uniform. This technique could be useful to concentrate particles at a drop surface, and also separate two types of particles. In this talk we show that in addition to the dielectrophoretic force the particles also interact with each other via the dipole-dipole interactions to form chains or move away from each other depending on the local direction of the electric field. The regions in which the local electric field is normal to the drop surface (poles), particles move away from each other. On the other hand, where the local direction of electric field is tangential to the drops surface (equator), they form chains that are aligned parallel to the electric field direction.

This work was supported by the grant R01-2009-0083830 from National Research Foundation (NRF) of Korea, and by the BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.

2:05PM CR.00006 Electrically Modulated Partial Coalescence of Oppositely Charged Droplets

J.C. CRESSEY, B.S. HAMLIN, W.D. RISTENPART, Dept. Chem. Engr. & Mat. Sci., Univ. California at Davis — Oppositely charged drops fail to coalesce above a critical field strength, despite the attractive force between the opposite charges. Here we report a technique to externally control the extent to which a charged droplet is allowed to coalesce. For sufficiently low ion conductivities, the degree of coalescence of water drops in oil can be tuned from complete coalescence at low field strengths to complete non-coalescence at high field strengths. Strikingly, in this regime the size and charge of the daughter droplet are both independent of the drop conductivity. We present evidence that the charge transfer is instead dominated by convective effects associated with the capillary-driven penetration of a vortex into the larger drop. Moreover, we demonstrate that measurements of the size of the daughter droplet are consistent with a model based on a balance between capillary forces and electrostatic repulsion.

2:18PM CR.00007 How Do Droplets Acquire Charge? W.D. RISTENPART, B.S. HAMLIN, Dept. Chem. Engr. & Mat. Sci., Univ. California at Davis — Liquid droplets immersed in a poorly conductive medium are known to acquire charge upon contact with an electrified surface. Although there is some evidence that electrochemical reactions limit the degree of charge transfer, the details of the underlying mechanism are unclear. Previous work has relied on estimates of the drop charge obtained from a balance between the electrophoretic driving force and viscous drag, which necessitates an accurate description of the hydrodynamic resistance of the drop. Here we establish a procedure to directly measure the droplet charge using a high resolution electrometer. Simultaneous high speed video and voltammetry provide a quantitative comparison of the drop charge obtained by the two methods. We find significant deviations between the measured charges and Maxwell’s limiting case for the charge acquired by a perfectly conducting rigid sphere in contact with a planar electrode. We interpret the discrepancy in terms of electrochemical limitations, and we provide physical evidence that under appropriate conditions electrophoretic reactions play a key role in the charge transfer to the drop.

2:31PM CR.00008 Formation and Electrically Induced Reversal of “Dynamic” Stagnant Caps

B.S. HAMLIN, W.D. RISTENPART, Dept. Chem. Engr. Mat. Sci. Univ. California at Davis — Drops are commonly observed to move more slowly than predicted by the classic Hadamard-Rybczyński model for the drag force on an immiscible spherical droplet. The discrepancy is commonly interpreted in terms of the presence of a “stagnant cap” of surfactant molecules at the trailing edge of the drop; the surfactants exert a Marangoni stress that impedes recirculation inside the drop. Here we present high-speed video of the formation and electrically induced reversal of “dynamic” stagnant caps. A charged water droplet is subjected to a sudden reversal in the direction of an applied electric field, and the overall motion of the drop and the relative motion of tracer particles on the droplet surface are observed. The droplet is shown to decelerate over a time period commensurate with the transient rearrangement of the tracer particles on the surface. We interpret the behavior in terms of the dynamic reversal of the stagnant cap, and we demonstrate that the observations are consistent with a scaling analysis of the transient cap rearrangement.

2:44PM CR.00009 Particle-particle Interactions on the Surface of a Drop Subjected to a Uniform Electric Field

SAI NUDURUPATI, MANSOOR JANJUA, Lake Superior State University, PUSHPENDRA SINGH, New Jersey Institute of Technology, NADINE AUBRY, Carnegie Mellon University — We recently proposed a technique in which an externally applied uniform electric field was used to alter the distribution of particles on the surface of a drop immersed in another immiscible liquid. Particles move along the drop surface to form a ring near the drop equator or collect at the poles depending on their dielectric constant relative to that of the two liquid involved. This motion is due to the dielectrophoretic force which acts upon particles because the electric field on the surface of the drop is non-uniform, despite the fact that the applied electric field is uniform. This technique could be useful to concentrate particles at a drop surface, and also separate two types of particles. In this talk we show that in addition to the dielectrophoretic force the particles also interact with each other via the dipole-dipole interactions to form chains or move away from each other depending on the local direction of the electric field. The regions in which the local electric field is normal to the drop surface (poles), particles move away from each other. On the other hand, where the local direction of electric field is tangential to the drops surface (equator), they form chains that are aligned parallel to the electric field direction.

Sunday, November 21, 2010 1:00PM - 3:10PM – Session CS Drops III: Evaporation

1:00PM CS.00001 Hydrodynamics of a sessile drop of capillary size, LEV BARASH, Landau Institute for Theoretical Physics, TERRY BIGIONI, James Frank Institute, The University of Chicago, VALERII VINOKUR, Material Science Division, Argonne National Laboratory, LEV SHCHUR, Landau Institute for Theoretical Physics — Several dynamical stages of the Marangoni convection of an evaporating sessile drop, effects of the thermal conduction in the drop and the diffusion of vapor in air. The stages are characterized by different number of vortices in the drop and the spatial location of vortices. During the early stage the array of vortices arises near a surface of the drop and induces a non-monotonic spatial distribution of the temperature over the drop surface. The number of near-surface vortices in the drop is controlled by the Marangoni cell size, which is calculated similar to that given by Pearson for flat fluid layers. The number of vortices quickly decreases with time, resulting in three bulk vortices in the intermediate stage. The vortex structure finally evolves into the single convection vortex in the drop, existing during about 1/2 of the evaporation time. Simulation results agree well with the data of evaporation rate measurements for the toluene drop. Computed dependence of contact angle of colloidal sessile droplet during evaporation coincide well with available experimental time dependence of angle of nanocrystal superlattice domains orientation.

This work was supported by the grant R01-2009-0083830 from National Research Foundation (NRF) of Korea, and by the BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.
1:13PM CS.00002 Contact Angle Hysteresis of an Evaporating Droplet¹, JEONGEUN RYU, SANGHYUN LEE, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — Contact angle and contact line dynamics of an evaporating droplet exhibit interesting features. During the evaporation process, contact angle decreases with time and becomes smaller than the static receding contact angle; and contact line is deformed and locally pinned. In this work, we attempted to explain the phenomenon in terms of contact angle hysteresis. We conjecture that impurities inside a droplet promote the heterogeneity of surface and induce the rapid decrease of contact angle lower than the receding contact angle. Based on our conjecture, we investigate experimentally and theoretically the effect of impurities as a source of contact angle hysteresis and subsequent change of contact angle.

¹This research was financially supported by a grant to MEMS Research Center for National Defense funded by Defense Acquisition Program Administration.

1:26PM CS.00003 Drying drops of blood¹, DAVID BRUTIN, BENJAMIN SOBAC, Université de Provence, BORIS LOQUET, Bio 13, JOSÉ SAPMOL, Université de la Méditerranée — The drying of a drop of human blood is fascinating by the complexity of the physical mechanisms that occur as well as the beauty of the phenomenon which has never been previously evidenced in the literature. The final stage of full blood evaporation reveals for a healthy person the same regular pattern with a good reproducibility. Other tests on anemia and hyperlipidemic persons were performed and presented different patterns. By means of digital camera, the influence of the motion of red blood cells (RBCs) which represent about 50% of the blood volume, is revealed as well as its consequences on the final stages of drying. The mechanisms which lead to the final pattern of dried blood drops are presented and explained on the basis of fluid and solid mechanics in conjunction with the principles of hematology. Our group is the first to evidence that the specific regular patterns characteristic of a healthy individual do not appear in a dried drop of blood from a person with blood disease. Blood is a complex colloidal suspension for which the flow motion is clearly non-Newtonian. When drops of blood evaporate, all the colloids are carried by the flow motion inside the drop and interact.

1:39PM CS.00004 Rush hour for particles suspended in drying drops, HANNEKE GELDERBLOM, ÁLVARO G. MARIN, JACCO H. SNOEIJER, DELTELF LOHSE, Physics of Fluids, University of Twente — In the late nineteen Deegan et al. explained the formation contact-line deposits in a drying sessile droplet suspension of particles (Nature 389 (1997), Physical Review E 60, (2000)). It was found that if there is evaporation from the drop edge while the contact line is pinned, liquid and particles are dragged towards the contact line creating the well known coffee-stain ring. Here, we analyze this process in detail by measuring the velocity field inside an evaporating droplet using µ-PIV. It was found that most of the particle transport occurs in the last moments of the droplet life-time. This rush explains the different characteristic packing of the particles in the layers of the ring, which is much more ordered in the thin outer part than in the thick inner one, since almost all particles arrive at the end. The rush-hour behavior of particles in evaporating drops can be attributed to the vanishing of the contact angle and follows from mass conservation.

1:52PM CS.00005 Surface wettability and triple line behavior controlled by nano-coatings: effects on the sessile drop evaporation¹, BENJAMIN SOBAC, DAVID BRUTIN, Université de Provence, JÉRÔME GAVILLET, CEA LITEN, JUSTI UMR CNRS 6595 TEAM, CEA LITEN TEAM — Sessile drop evaporation is a phenomenon commonly came across in nature or in industry with cooling, paintings or DNA mapping. However, the evaporation of a drop posed on a substrate is not completely understood due to the complexity of the problem. Here we investigate, with several nano-coating of the substrate (SiOx, SiOc and CF), the wettability and the triple line dynamic of a sessile drop under natural phase change. The experiment consists in analyzing simultaneously the kinetics of evaporation, internal thermal motion and heat and mass transfer. Measurements of temperature, heat-flux and visualizations with visible and infrared cameras are performed. The dynamic of the evaporative heat flux appears clearly different for a drop evaporating in pinned mode than in receding mode. Moreover, the kinetics of evaporation, the internal flow structure and the evaporative heat flux are drastically influenced by the wettability the substrate.

¹The authors acknowledge the support of the “French National Research Agency” (Agence Nationale de la Recherche) via the grant ANR-09-BLAN-0093-3.

2:05PM CS.00006 Diffusion-Controlled Evaporating Stationary Meniscus in a Channel, JEAN-PIERRE NJANTE, STEPHEN MORRIS, UC Berkeley — Isochemical liquid evaporates into a mixture of its own vapor and an inert component. On one wall, the contact line is pinned; the other wall is perfectly wetted. These walls are at uniform temperature $T_o$ equalling that of the distant gas. Liquid evaporates because the partial pressure $p_{\infty}$ of the distant vapor is less than the saturation pressure $P$ evaluated at $T_o$ and pressure $P_{\infty}$ of the distant liquid. Evaporation draws liquid into the contact region; near the wetted wall, the resulting pressure differences distorts the interface, creating an apparent contact angle. $\theta$ is a flow property and increases with the control parameter $P(p_{\infty}, T_o) - p_{\infty}$. As a preliminary to finding $\theta$, we prove the following: (a) The system is effectively isothermal; though evaporation induces liquid temperature differences, they are kinetically negligible. (b) Whenever the continuum approximation holds within the gas, diffusion is rate-limiting. As a result, liquid and vapor at the interface are in local thermodynamic equilibrium; the vapor partial pressure is related to liquid pressure by Kelvin’s equation $P^v = P(p^v, T_o)$. Given (a) and (b), the film thickness $h(x,y)$, is determined by a system comprising of the steady state diffusion equation for $p^v(x,y)$, the lubrication equation for $p^v(x)$, and the augmented Young-Laplace equation for $h$. These equations are coupled by Kelvin’s equation. We use our solution to address the corresponding problem for a droplet on a substrate.

2:18PM CS.00007 An evaporation model of multicomponent solution drops, SILVANA SARTORI, Mech. and Aerospace Eng Dept. University of California San Diego, AMABLE LIÑÁN, Escuela Técnica Superior de Ingenieros Aeronáuticos. UPM (Spain), JUAN C. LASHERAS, Mech. and Aerospace Engr Dept. UC San Diego — Solutions of polymers are widely used in the pharmaceutical industry as tablets coatings. These allow controlling the rate at which the drug is delivered, taste or appearance. The coating is performed by spraying and drying the tablets at moderate temperatures. The wetting of the coating solution on the pill’s surface depends on the droplet Webber and Re numbers, angle of impact and on the rheological properties of the droplet. We present a model for the evaporation of multicomponent solutions droplets in a hot air environment with temperatures substantially lower than the boiling temperature of the solvent. As the liquid vaporizes from the surface the fluid in the drop increases in concentration, until reaching its saturation point. After saturation, precipitation occurs uniformly within the drop. As the surface regresses, a compacting front formed by the precipitate at its maximum packing density advances into the drop, while the solute continues precipitating uniformly. This porous shell grows fast due to the double effect of surface regression and precipitation. The evaporation rate is determined by the rates at which heat is transported to the droplet surface and at which liquid vapor diffuses away from it. When the droplet is fully compacted, the evaporation is drastically reduced.
The deposition of bi-dispersed particles in inkjet-printed evaporating colloidal drops. YING SUN, ABHIJIT JOSHI, VIRAL CHHASATIA, Drexel University — In this study, the deposition behaviors of inkjet-printed evaporating colloidal drops consisting of bi-dispersed micro and nano-sized particles are investigated by fluorescence microscopy and SEM. The results on hydrophilic glass substrates show that, evaporatively-driven outward flow drives the nanoparticles to deposit close to the pinned contact line while an inner ring deposition is formed by microparticles. This size-induced particle separation is consistent with the existence of a wedge-shaped drop edge near the contact line region of an evaporating drop on a hydrophilic substrate. The replenishing evaporatively-driven flow assembles nanoparticles closer to the pinned contact line forming an outer ring of nanoparticles and this particle jamming further enhances the contact line pinning. Microparticles are observed to form an inner ring inside the nano-sized deposits. This size-induced particle separation presents a new challenge to the uniformity of functional materials in bioprinting applications where nanoparticles and micro-sized cells are mixed together. On the other hand, particle self-assembly based on their sizes provides easy and well-controlled pattern formation. The effects of particle size contrast, particle volume fraction, substrate surface energy, and relative humidity of the printing environment on particle separation are examined in detail.

Evaporation of a sessile droplet: Inside the coffee stain. ANNA HOANG, GUILLAUME BERTELOOT, UCLA, ADRIAN DAERR, MSc, Paris 7, PIROUZ KAVEHPOUR, UCLA, FRANCOIS LEQUEUX, ESPCI, Paristech, LAURENT LIMAT, MSc, Paris 7 — The deposition of uniform layers of colloids on a solid surface is a major challenge for several industrial processes such as glass surface treatment and creating optical filters. A possible strategy involves the deposition of the colloids behind a contact line that recedes due to hydrodynamic reasons and evaporation (drying). We have investigated a drop of colloidal suspension evaporating on a flat surface where the contact line remains strongly pinned on the surface. We have observed that the deposit grows from the contact line following a $r^{-1/2}$ law and then accelerates with surprising spatial and temporal modulations. The power law can be recovered by a ballistic model, in which the particles are driven to contact line by the evaporation field that diverges near the contact line.

Deposits of drying drops of a nanotube suspension. MINIY VARGAS, CALEB LIMON, OSCAR SARMIENTO, Instituto Tecnologico de Zacatepec, GUILLERMO HERNANDEZ-CRUZ, EDUARDO RAMOS, MARINA RINCON, Universidad Nacional Autonoma de Mexico, DEPARTAMENTO METALMECANICA COLLABORATION, CENTRO DE INVESTIGACION EN ENERGIA COLLABORATION — We have made observations of the pattern formed by deposits of an evaporating sessile drop of a carbon nanotube suspension. The nanotubes are chains of carbon molecules, 2 nm diameter and approximately 15 micrometers long. The suspension concentration is 0.25 mg/ml and initially, the droplet volume is 2 µl. Nanotubes are transported by the flow generated by evaporation of the drop and the resulting patterns are the result of the drag of the filaments by the fluid motion. The pattern observed is composed of a circular band with several (order ten) spots with higher concentration of nanotubes. Also, the inner ring of the band displays a higher concentration of nanotubes. At contrast to similar observations where the suspensions are prepared with microspheres, no ring formation at the outer edge of the initial footprint of the drop (coffee effect) is clearly identified. Our observations are interpreted in terms of existing theories.

Organisms that evolve within complex fluidic environments often develop adaptive behaviors that are in response to these environments. Here we report experimental observations on the locomotion of C. elegans swimming in arrays of micro-pillars in square lattices, with different lattice spacing, and in structured environments, is ubiquitous in nature. They navigate complex environments consisting of fluids and obstacles, negotiating hydrodynamic effects and geometrical constraints. Here we report experimental observations on the locomotion of C. elegans swimming in arrays of micro-pillars in square lattices, with different lattice spacing. We observe that the worm employs a number of different locomotion strategies depending on the lattice spacing. As observed previously in the literature, we uncover regimes of enhanced locomotion, where the velocity is much higher than the free-swimming velocity. In addition, we also observe changes in the swimming regime as a function of lattice spacing. We also track the worm over time and find that it exhibits super-diffusive behavior and covers a larger area by utilizing the obstacles. These results may have significant impact on the foraging behavior of the worm in its natural environment. Our experimental approach, in conjunction with modeling and simulations, allows us to disentangle the effects of structure and hydrodynamics for an undulating microorganism.

The locomotion of C. elegans in structured environments. TRUSHANT MAJMUDAR, ERIC KEAVENY, MICHAEL SHELLEY, JUN ZHANG, Courant Institute, New York University — Undulatory locomotion of microorganisms like soil-dwelling worms and sperm, in structured environments, is ubiquitous in nature. They navigate complex environments consisting of fluids and obstacles, negotiating hydrodynamic effects and geometrical constraints. Here we report experimental observations on the locomotion of C. elegans swimming in arrays of micro-pillars in square lattices, with different lattice spacing. We observe that the worm employs a number of different locomotion strategies depending on the lattice spacing. As observed previously in the literature, we uncover regimes of enhanced locomotion, where the velocity is much higher than the free-swimming velocity. In addition, we also observe changes in the swimming regime as a function of lattice spacing. We also track the worm over time and find that it exhibits super-diffusive behavior and covers a larger area by utilizing the obstacles. These results may have significant impact on the foraging behavior of the worm in its natural environment. Our experimental approach, in conjunction with modeling and simulations, allows us to disentangle the effects of structure and hydrodynamics for an undulating microorganism.

Simulations of C. elegans locomotion through a structured medium. ERIC KEAVENY, TRUSHANT MAJMUDAR, JUN ZHANG, MICHAEL SHELLEY, Courant Institute, New York University — The small nematode C. elegans serves as a model system for studies in both biology and engineering. Using a realistic Reynolds number undulatory locomotion, particularly in fluids with an embedded microstructure that is comparable in size to the swimmer. Recent experimental observations of C. elegans locomotion in a lattice of obstacles indicate that the worm can achieve speeds as much as an order of magnitude greater than its free-swimming value. In addition to a series of experimental studies of this phenomenon, we perform numerical simulations of a self-locating chain of beads in a lattice of spherical obstacles. We explore the dependence of the worm's speed on the frequency of undulation and lattice spacing and quantify the necessary conditions for enhanced locomotion. We also use the simulations to characterize the forces experienced by the worm in this regime. Further, by comparing the simulation results with our experimental data, we identify changes in worm locomotive behavior in response to imposed geometric conditions.

Locomotion and Body Shape Changes of Metabolically Different C.elegans in Fluids with Varying Viscosities. RACHEL WONG, University of Washington, NOAH BRENOWITZ, NYU, AMY SHEN, University of Washington — Caenorhabditis elegans (C.elegans) are soil dwelling roundworms that have served as model organisms for studying a multitude of biological and engineering phenomena. On agar, the locomotion of the worm is sinusoidal, while in water, the swimming motion of the worm appears more episodic. The efficiency of the worm locomotion is tested by placing the worm in four fluids with varying viscosities. We quantify the locomotion pattern variations by categorizing the swimming kinematics and shapes of the C.elegans. The locomotion of two mutants C.elegans and a control C.elegans was tested: daf-2, nhr49, and jv2 Wildtype. The metabolic effects of the mutants are evaluated by focusing on the forward swimming velocity, wavelength, amplitude and swimming frequency were compared. Using these measured values, we were able to quantify the efficiency, the speed of propagation of the wave along the body resulting in forward movement (wave velocity), and transverse velocity, defined as the amplitude times the frequency, of the worm locomotion. It was shown that C.elegans has a preferential swimming shape that adapts as the environment changes regardless of its efficiency.

Motility of small nematodes in disordered wet granular media. GABRIEL JUAREZ, KEVIN LU, JOSUE SZNITMAN, PAULO E. ARRATIA, University of Pennsylvania — Organisms that evolve within complex fluidic environments often develop unique methods of locomotion that allow them to exploit the properties of the media. In this talk, we present an investigation on the motility of the worm nematode Caenorhabditis elegans in shallow, wet granular media as a function of particle size dispersity and area density (d) using both particle- and nematode-based tracking methods. Surprisingly, the nematode's propulsion speed is enhanced by the presence of particles in a fluid and is nearly independent of local area density. The undulation speed, often used to differentiate locomotion gaits, is significantly affected by particle size dispersity for area densities above $d > 0.55$, and is characterized by a change in the nematode's waveform from swimming to crawling. This change occurs for dense polydisperse media only and highlights the organism's adaptability to subtle differences in local structure between monodisperse and polydisperse media.

2:57PM CS.00010 Deposits of drying drops of a nanotube suspension.

Sunday, November 21, 2010 1:00PM - 2:57PM
Session CT Biolocomotion II: Slithering, Crawling and Undulatory Swimming
Long Beach Convention Center Grand Ballroom B

1:00PM CT.00001 Locomotion of C. elegans in structured environments. TRUSHANT MAJMUDAR, ERIC KEAVENY, MICHAEL SHELLEY, JUN ZHANG, Courant Institute, New York University — C. elegans swimming in arrays of micro-pillars in square lattices, with different lattice spacing.
Collective behavior of nematodes in a thin fluid, SEAN GART, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University — Many organisms live in a confined fluidic environment such as in a thin fluid layer on dermal tissues, in saturated soil, and others. In this study, we investigate collective behaviors of nematodes in a thin fluid layer. The actively moving nematodes feel various hydrodynamic forces such as surface tension from the top air-liquid interface, viscous stress from the bottom surface, and more. Two or more nematodes in close proximity can be drawn together by the capillary force between bodies. This capillary force also makes it difficult for nematodes to separate. The Strouhal number and a ratio of amplitude to wavelength are measured before and after nematode aggregation and separation. Grouped and separate nematodes have no significant changes of the Strouhal number and the ratio of amplitude to wavelength, which shows that body stroke and kinematic performance do not change while grouped together. This result implies that nematodes gain no mechanical advantage during locomotion when grouped but that the capillary force draws and keeps nematodes joined together.

The Effect of Electric Field Magnitude and Frequency on Caenorhabditis Elegans, HAN-SHENG CHUANG, DAVID RAIZEN, NOOREEN DABBISH, ANNESAI LAMB, HAIM BAU, University of Pennsylvania — Low magnitude, DC electric fields have been used to guide the motion of the wild-type nematode (worm) Caenorhabditis elegans. Low intensity AC fields (< 100 Hz) can even be utilized to localize the worm. However, the worm appears oblivious to the electric field as the frequency is higher than several hundreds of Hz. In contrast, in the presence of nonuniform, moderate AC fields (> 10 Hz), the worm is subject to an undulatory swimming motion that results from the first demonstration of dielectrophoretic trapping of an animal. With certain electrode arrangements, only the worm’s tail is immobilized, and the worm’s swimming motion does not appear to be affected by the trapping force. Similar trapping conditions with transitional frequencies (~10–100 kHz) can cause paralysis. The worm is (irreversibly) paralyzed with lower frequencies (e.g. 45 kV/m, 2 kHz) or electrified with higher electric field intensities (e.g. 10 Hz, 70 kV/m). We report on the results of a parametric study that delineates the effect of the electric field on the worm as a function of the worm’s stage and the electric field intensity and frequency. Worm-dielectrophoresis can be used, among other things, to sort worms by size, to temporarily immobilize worms to enable their characterization and study, and to use worms to induce fluid motion and mixing.

Could gastropods crawl using Newtonian mucus? JANICE LAI, Stanford University, MARIA VAZQUEZ-TORRES, UCIII Madrid, JUAN C. DEL ALAMO, UC San Diego, JAVIER RODRIGUEZ-RODRIGUEZ, UCIII Madrid, JUAN C. LASHERAS, UC San Diego — The locomotion of terrestrial gastropods is driven by a train of periodic muscle contractions (pedal waves) and relaxations (interwaves) that propagate from their tail to their head (direct waves). We study the locomotion of these animals on smooth flat surfaces by measuring the three-dimensional displacements of the ventral foot surface induced by the passage of the waves. A simple model based on lubrication theory is proposed in accordance with the experimental observations. This model uncover a new mode of locomotion that works even when the lubricant between the foot and the animal is Newtonian. The model can also be adapted to situations where the animal’s foot is in contact with the ground only at discrete points, as is the case when it crawls on a wire mesh or on rough soil surfaces. Furthermore, comparison between the stress exerted by the animal on the substrate and the model predictions allows us to clarify the role of the complex rheology observed in the mucus of terrestrial gastropods.

Concertina locomotion of snakes, HAMIDREZA MARVI, DAVID HU, Georgia Institute of Technology — Snake-like modes of locomotion may easily traverse water as well as land. In this combined experimental and theoretical investigation, we investigate the accordion-like modes of locomotion, in which snakes move by a series of extensions and contractions of their bodies. Snakes are filmed performing concertina locomotion on flat cloth surfaces arranged at various angles of inclination. Using the body kinematics of the snakes and friction properties of their skin, we model snake locomotion using a three-mass model propelled by sliding friction. Particular attention is paid to maximizing propulsive efficiency using optimum rates of contraction and snake scales as braces for ascending inclines.

Comparison of physical, numerical and resistive force models of undulatory locomotion within granular media, DANIEL I. GOLDMAN, RYAN D. MALADEN, YANG DING, Georgia Institute of Technology, PAUL UMBANHOWAR, Northwestern University — We integrate biological experiments, empirical theory, numerical simulation, and a physical robot model to reveal principles of undulatory locomotion in granular media. High speed x-ray imaging of the sandfish, Scincus scincus, in 3 mm glass particles reveals that it swims within the medium without limb use by propagating a single period traveling sinusoidal wave down its body, resulting in a wave efficiency, η, the ratio of its average forward speed to wave speed, of 0.51 ± 0.13. A resistive force theory (RFT) which balances granular thrust and drag forces along the body predicts η close to the observed value. We test this prediction against two other modeling approaches: a numerical model of the sandfish coupled to a Molecular Dynamics (MD) simulation of the granular medium, and an undulatory robot which swims within granular media. We use these models and analytic solutions of the RFT to vary the ratio of undulation amplitude to wavelength (A/λ) and demonstrate an optimal condition for sand-swimming that results from competition between η and λ. The RFT, in agreement with simulation and robot models, predicts that for a single period sinusoidal wave, maximal speed occurs for A/λ ≈ 0.2, the same kinematics used by the sandfish.

Phase field method for interfacial fluid flow with soluble surfactant, MINH DO-QUANG, Research Associate, STEFAN ENGELBLOM, Postdoc, ANNA-KARIN TORNBERG, Associate Professor, GUSTAV AMBERG, Professor, LINNE FLOW CENTER, MECHANICS DEPT., ROYAL INSTITUTE OF TECHNOLOGY, STOCKHOLM, SWEDEN TEAM, LINNE FLOW CENTER, CSC/NA, ROYAL INSTITUTE OF TECHNOLOGY, STOCKHOLM, SWEDEN TEAM — In this study, a simulation of flow of two immiscible fluids with a surfactant is studied using a diffuse interface formulation. The finite element method with adaptive mesh refinement is used to solve the Navier-Stokes equations together with the phase field equation. This system allows us to simulate the motion of a free surface in the presence of surface tension and the effect of surfactant. The method is valid for simple test cases and the computational results are found to be in a good agreement with the analytical solutions. The method is then being used to study the effect of surfactant on rippled deformation of buoyancy-dribbled bubbles, and drop breakup and coalescence in a circular tube. We also discuss the free energy used in this approach and some ways to improve it.

Computer time provided by SNIC (Swedish National Infrastructure for computing) is gratefully acknowledged.

Strategies for Multiphase Flows with High Density Ratios, OLIVIER DESJARDINS, University of Colorado at Boulder, VINCENT MOUREAU, CORIA, France — While numerical methods for multiphase flows have progressed significantly in the past few years, simulating realistic flows with high density ratios remains a major hurdle, especially when combined with high shear, as encountered in air-blast atomization devices. In order to alleviate this issue in the context of level set methods, two strategies are investigated that aim at improving the consistency between level set and momentum transport. The first strategy relies on transporting an auxiliary density field created from the level set function and using it for creating consistent momentum fluxes. The second strategy relies on a two-velocity ghost fluid approach where both gas and liquid velocities are considered, between level set and momentum transport. The second strategy is the one that we have explored in this study and that we will present in this paper. We have shown that this strategy is able to significantly improve the consistency between level set and momentum transport, especially for realistic fuel injection applications.
1:26PM CU.00003 A Sub-Grid Model for Surface Tension Induced Phase Interface Dynamics\textsuperscript{1} , MARCUS HERRMANN, Arizona State University — In many flows involving liquid/gas phase interfaces, small scale interface dynamics play an important role. In atomization of liquids, for example, surface tension forces can dominate the final stages of topology change events. Resolving such surface tension dominated small scale interface dynamics in a flow solver quickly becomes prohibitively expensive, especially for turbulent flows of engineering relevance. However, filtering the governing equations introduces unclosed terms that require modeling, among them the filtered surface tension force. In single phase flows, models, like the Large Eddy Simulation (LES) approach, rely on the existence of a cascade process, which is not necessarily present in two phase flows with surface tension forces. We thus propose a novel sub-grid model for the surface tension force based on the Refined Level Set Grid method that does not imply the existence of a cascade process. It is based on a local Taylor analogy, with surface tension acting as a local spring and viscosity as a local damper. We present results for the sub-grid motion of oscillating drops and sub-grid Rayleigh-Plateau instabilities commonly encountered in turbulent atomization.

\textsuperscript{1}This work was supported by NSF grant CBET-0853627 and the Center for Turbulence Research 2010 Summer Program.

1:39PM CU.00004 Contact-line motion past sharp corners: from spreading to jetting , PETER SPELT, Department of Chemical Engineering, Imperial College London, YI SUI, Department of Chemical Engineering, Imperial College London — A level-set method for the simulation of two-phase flows with moving contact lines has been adapted to simulate contact-line motion past sharp corners numerically. The method is not restricted to creeping-flow regimes, and alleviates the stress singularity at a moving contact line by the use of a slip condition. First, the detailed flow behaviour is studied at the instance when the contact line moves past a corner, as well as at that at later times, to see to what extent this deviates from conventional spreading behaviour. The method is then used to investigate the motion of a liquid out of an injection channel. Several flow regimes are observed, including a transition from spreading to jetting.

1:52PM CU.00005 A Numerical Approach for Simulating Incompressible Two-Phase Flows Considering Surface Tension Effect\textsuperscript{1} , SANG HUN CHOI, MYUNG HWAN CHO, Seoul National University, HYOU NG GWON CHOI, Seoul National University of Technology, JUNG YUL YOO, Seoul National University — A novel level set method is proposed to simulate the incompressible two-phase flow considering the effect of surface tension. A mixed element is adopted, so that the continuity and Navier-Stokes equations are solved by using the Q2Q1 integrated finite element method, and the level set function is solved by using the Q1Q1 finite element method. For reinitialization of level set function, a direct approach method is employed, instead of solving hyperbolic type equation. In order to verify the accuracy and robustness of the code, the present method is applied to a few benchmark problems including the Rayleigh instability, bubble rising, and bubble breaking problems. It is confirmed that the present results are in good qualitative and quantitative agreements with the existing studies.

\textsuperscript{1}The authors are supported by the Korea Research Foundation Grant funded by the Korean Government (MOEHRD) (KRF-0420-20070058).

2:05PM CU.00006 A mass-conserving volume of fluid method for two-phase incompressible isotropic turbulence , ALBERTO BARALDI, ANTONINO FERRANTE, University of Washington, Seattle — We implemented and investigated a volume of fluid (VoF) method for capturing the motion of an initially spherical interface in incompressible velocity fields. First, we tested the interface reconstruction and advection algorithms in analytical velocity fields: linear translation, solid body rotation, and Taylor-Green vortex. These tests showed that the implemented VoF method conserves mass with machine accuracy. Then, we tested the VoF in incompressible isotropic turbulence. In order to compute the geometrical error, the instantaneous velocity field extracted from DNS was artificially reversed in time by means of a cosine time-function. During this test, the spherical interface, with a diameter of Taylor-length-scale size, deforms, breaks, reconnects and returns to its initial position. Also in this test, our results show that the VoF method conserves mass with machine accuracy. Furthermore, the topology changes are captured without any ad hoc treatment. Thus, we conclude that the implemented VoF is suitable for DNS of two-phase (e.g. gas-liquid or liquid-liquid) incompressible isotropic turbulence.

2:18PM CU.00007 A volume-of-fluid interfacial flow solver with advected normals , MEHDI RAESSI, University of Massachusetts-Dartmouth, JAVAID MOSTAGHMI, MARKUS BUSSMANN, University of Toronto, UNIVERSITY OF MASSACHUSETTS-DARTMOUTH TEAM, UNIVERSITY OF TORONTO TEAM — We introduce an implementation of the adverting normals method in a volume-of-fluid interfacial flow solver. The advected normals are used to compute the interface curvature for calculating the surface tension force, and for reconstructing the interface in a volume-conserving volume-of-fluid method. To improve the performance of the method in under-resolved regions of the flow, where normals vary sharply, a curvature-based criterion is used to detect and correct poorly defined normals. We present results of advection as well as actual flow problems and demonstrate that the new method is well suited for problems that involve large interface deformation and breakup.

2:31PM CU.00008 Numerical simulation of two-phase flows in complex geometries by combining two different immersed-boundary methods , BO YIN, HAOXIANG LUO, Vanderbilt University — Two-phase flows in various industrial applications often occur in complex geometries. To simulate this type of flows, we have combined two different immersed-boundary methods to handle the fluid-fluid and the fluid-solid interfaces separately. For the fluid-fluid interface, a diffuse-interface method is employed where the discontinuities of the material properties and the traction jump are all regularized using an approximate Dirac’s Delta function. For the fluid-solid interface, ghost nodes and a local flow reconstruction are employed to complement the finite-difference discretization and to incorporate the boundary conditions. A single-block Cartesian mesh is used to discretize the entire domain. Both 2D and 3D codes have been implemented. Validation and code applications will be demonstrated at the conference. \textsuperscript{*}Supported by the ACS Petroleum Research Fund.

2:44PM CU.00009 Multi-scale modeling of compressible multi-fluid flow with sharp-interface method , XIANGYU HU, NI KOLAUS ADAMS, Technical University of Munich — One important issue associated with the complexity of the dynamically evolving material interface is the scale- or mixing-dependent dynamics, which suggests very different physical phenomena depending on resolution and mixing of the material interface. We would like to present an idea of multi-scale modeling, in which the interface interaction is modeled as mechanical non-equilibrium or equilibrium depending on the scale measurement or resolvability of the interface. The work based on my previously developed conservative sharp-interface method for compressible multi-phase flows. In this finite-volume method, the interface is presented by a level-set function and the interface interaction is modeled by solving two-material Riemann problem. To extend this method for multi-scale modeling, two important issues have been addressed: one is a scale-separation algorithm for identifying the resolved and unresolved interface; the other is to a mechanical equilibrium model and its coupling to the sharp-interface model with simple and efficient approaches.

2:49PM CU.00010 Efficient implementation of a multi-scale multi-fluid pressure solver , KIYOKAZU TANIGUCHI, University of Massachusetts, AMIR RATTAN, University of Massachusetts — We present a class of efficient multi-scale multi-fluid flow solvers that are based on the MUSCL adaptive strengthENO scheme, and a pressure solver that is based on a parallel artificial boundary condition (ABC) method. The multi-scale solver is used to model the flow of different species with different diffusivities, and the parallel ABC method is used to simulate the flow of different species in the presence of a wall. The two methods are used to model a two-phase flow of a fluid and a solid. The multi-scale solver is used to model the flow of a fluid and a solid in the presence of a wall. The parallel ABC method is used to simulate the flow of a fluid and a solid in the presence of a wall.
1:00PM CV.00001 Particle pair dispersion in turbulent boundary layer, CRISTIAN MARCHIOLI, ENRICO PITTON, ALFREDO SOLDATI, Dept. Energy Technologies, University of Udine, FEDERICO TOSCHI, Dept. Applied Physics, Technische Universität Eindhoven — The rate at which two particles separate in turbulent flow is of central importance to predict the spatial distribution of inhomogeneities and to characterize mixing. Pair separation is analyzed for the specific case of small inertial particles in dispersed turbulent channel flow to determine the role of mean shear and small-scale structures. To this aim an Eulerian-Lagrangian approach based on pseudo-spectral direct numerical simulation of fully-developed gas-solid flow at friction Reynolds number $Re_{f}=150$ is used. Pair separation statistics were computed for particles with different inertia released from different regions of the channel. Results demonstrate (i) that shear-induced effects predominate in the near-wall region, where velocity gradients reach a maximum, whereas small-scale fluctuations predominate away from the wall, where turbulence becomes more homogeneous and isotropic; and (ii) that the modalities by which particles become affected depend strongly on inertia. Starting from these results, open modelling issues will be addressed.

1:13PM CV.00002 Self-ordering of microscopic particles in time-periodic flows, DMITRI PUSHKIN, DENIS MELNIKOV, VALENTINA SHEVTSOVA, University of Brussels (ULB) — Small macroscopic particles advected by fluid flows are generally believed to follow the surrounding fluid when their Stokes number, $St ≪ 1$. We show that even when particles are as small as $St ≈ 10^{-6}$, the inertia effects may lead to spontaneous self-organization of particles into dynamic coherent structures. The arising structures are typically spiral curves. They are a collective phenomenon observed when an ensemble of particles is evolved. While the structures are robust for a range of control parameters, they become sensitive quickly dissolve and particles mix with the fluid outside this range. We explain the structures’ formation by the dynamical effect of phase-locking. It occurs for particle turn-over motions in vortical time-periodic flows. We show that this mechanism is responsible for the surprising assembly of particles into rotating spirals that was discovered experimentally in thermodiffusional flows more than a decade ago and has remained unexplained until now. In our exposition we lean on the results of our numerical simulations, which reproduce the effect in physically realistic regimes. We expect that similarly to phase-locking in dynamical systems, this effect is subtle but generic and may cause localization and ordering of particles in time-periodic flows that abound in nature and applications.

1:26PM CV.00003 Effects of turbulence intensity and gravity on transport of inertial particles across a shearless turbulence interface, GARRETT GOOD, SERGIY GERASHCHENKO, ZELLMAN WARHAFT, Cornell University — Water droplets of sub-Kolmogorov size are sprayed into the turbulence side of a shearless turbulent-non-turbulent interface (TNI) as well as a turbulent-turbulent interface (TTI). An active grid is used to form the mixing layer and a splitter plate separates the droplet-non droplet interface near the origin. Particle concentrations downstream are determined by Phase Doppler Particle Analyzer, the velocity field by hot wires, and the droplet accelerations by particle tracking. As for a passive scalar, for the TTI, the concentration profiles are described by an error function. For the TNI, the concentration profiles fall off more rapidly than for the TTI due to the large-scale intermittency. The profile evolution and effects of initial conditions are discussed, as are the relative importance of the large and small scales in the transport process. It is shown that the concentration statistics are better described in terms of the Stokes number based on the large scales than the small, but some features of the mixing are determined by the small scales, and these will be discussed. Sponsored by the U.S. NSF.

1:39PM CV.00004 Conditional statistics of inertial particle entrainment across a shearless turbulent interface, SERGIY GERASHCHENKO, GARRETT GOOD, ZELLMAN WARHAFT, Cornell University — For the turbulent-non-turbulent shearless mixing layer described by Good et al. ("Effects of turbulence intensity and gravity on transport of inertial particles across a shearless turbulence interface," G. Good, S. Gerashchenko, Z. Warhaft, APS, DFD 2010), the large scales on the turbulent side showed a strong influence on the particle dynamics in the intermittent region of the mixing layer. Particle statistics conditioned on the turbulent bursts were measured and compared with the unconditioned statistics. The un-conditional statistics adequately describe the particle transport across the mixing layer. The conditional statistics show changes in particle concentration within the bursts as a function of penetration into the mixing layer and this is discussed in terms of the particle history. The effect of gravity on the conditioned particle dynamics, in particular on the time of arrival statistics and radial distribution function, is addressed. Sponsored by the U.S. NSF.

1:52PM CV.00005 Inertial particles in a shearless mixing layer: direct numerical simulations, PETER IRELAND, LANCE COLLINS, Cornell University — Entrainment, the drawing in of external fluid by a turbulent flow, is present in nearly all turbulent processes, from exhaust plumes to oceanic thermoclines to cumulus clouds. While the entrainment of fluid and of passive scalars in turbulent flows has been studied extensively, comparatively little research has been undertaken on inertial particle entrainment. We explore entrainment of inertial particles in a shearless mixing layer across a turbulent-non-turbulent interface (TNI) and a turbulent-turbulent interface (TTI) through direct numerical simulation (DNS). Particles are initially placed on one side of the interface and are advanced in time in decaying turbulence. Our results show that the TTI is more efficient in mixing droplets than the TNI. We also find that without the influence of gravity, over the range of Stokes numbers present in cumulus clouds, particle concentration statistics are essentially independent of the dissipation scale Stokes number. The DNS data agrees with results from experiments performed in a wind tunnel with close parametric overlap. We anticipate that a better understanding of the role of gravity and turbulence in inertial particle entrainment will lead to improved cloud evolution predictions and more accurate climate models. Sponsored by the U.S. NSF.

Sunday, November 21, 2010 1:00PM - 3:10PM –
Session CV Particle Laden Flows II
Hyatt Regency Long Beach
Regency B

1:00PM CV.00001 Particle pair dispersion in turbulent boundary layer, CRISTIAN MARCHIOLI, ENRICO PITTON, ALFREDO SOLDATI, Dept. Energy Technologies, University of Udine, FEDERICO TOSCHI, Dept. Applied Physics, Technische Universität Eindhoven — The rate at which two particles separate in turbulent flow is of central importance to predict the spatial distribution of inhomogeneities and to characterize mixing. Pair separation is analyzed for the specific case of small inertial particles in dispersed turbulent channel flow to determine the role of mean shear and small-scale structures. To this aim an Eulerian-Lagrangian approach based on pseudo-spectral direct numerical simulation of fully-developed gas-solid flow at friction Reynolds number $Re_{f}=150$ is used. Pair separation statistics were computed for particles with different inertia released from different regions of the channel. Results demonstrate (i) that shear-induced effects predominate in the near-wall region, where velocity gradients reach a maximum, whereas small-scale fluctuations predominate away from the wall, where turbulence becomes more homogeneous and isotropic; and (ii) that the modalities by which particles become affected depend strongly on inertia. Starting from these results, open modelling issues will be addressed.

1:13PM CV.00002 Self-ordering of microscopic particles in time-periodic flows, DMITRI PUSHKIN, DENIS MELNIKOV, VALENTINA SHEVTSOVA, University of Brussels (ULB) — Small macroscopic particles advected by fluid flows are generally believed to follow the surrounding fluid when their Stokes number, $St ≪ 1$. We show that even when particles are as small as $St ≈ 10^{-6}$, the inertia effects may lead to spontaneous self-organization of particles into dynamic coherent structures. The arising structures are typically spiral curves. They are a collective phenomenon observed when an ensemble of particles is evolved. While the structures are robust for a range of control parameters, they become sensitive quickly dissolve and particles mix with the fluid outside this range. We explain the structures’ formation by the dynamical effect of phase-locking. It occurs for particle turn-over motions in vortical time-periodic flows. We show that this mechanism is responsible for the surprising assembly of particles into rotating spirals that was discovered experimentally in thermodiffusional flows more than a decade ago and has remained unexplained until now. In our exposition we lean on the results of our numerical simulations, which reproduce the effect in physically realistic regimes. We expect that similarly to phase-locking in dynamical systems, this effect is subtle but generic and may cause localization and ordering of particles in time-periodic flows that abound in nature and applications.

1:26PM CV.00003 Effects of turbulence intensity and gravity on transport of inertial particles across a shearless turbulence interface, GARRETT GOOD, SERGIY GERASHCHENKO, ZELLMAN WARHAFT, Cornell University — Water droplets of sub-Kolmogorov size are sprayed into the turbulence side of a shearless turbulent-non-turbulent interface (TNI) as well as a turbulent-turbulent interface (TTI). An active grid is used to form the mixing layer and a splitter plate separates the droplet-non droplet interface near the origin. Particle concentrations downstream are determined by Phase Doppler Particle Analyzer, the velocity field by hot wires, and the droplet accelerations by particle tracking. As for a passive scalar, for the TTI, the concentration profiles are described by an error function. For the TNI, the concentration profiles fall off more rapidly than for the TTI due to the large-scale intermittency. The profile evolution and effects of initial conditions are discussed, as are the relative importance of the large and small scales in the transport process. It is shown that the concentration statistics are better described in terms of the Stokes number based on the large scales than the small, but some features of the mixing are determined by the small scales, and these will be discussed. Sponsored by the U.S. NSF.

1:39PM CV.00004 Conditional statistics of inertial particle entrainment across a shearless turbulent interface, SERGIY GERASHCHENKO, GARRETT GOOD, ZELLMAN WARHAFT, Cornell University — For the turbulent-non-turbulent shearless mixing layer described by Good et al. ("Effects of turbulence intensity and gravity on transport of inertial particles across a shearless turbulence interface," G. Good, S. Gerashchenko, Z. Warhaft, APS, DFD 2010), the large scales on the turbulent side showed a strong influence on the particle dynamics in the intermittent region of the mixing layer. Particle statistics conditioned on the turbulent bursts were measured and compared with the unconditioned statistics. The un-conditional statistics adequately describe the particle transport across the mixing layer. The conditional statistics show changes in particle concentration within the bursts as a function of penetration into the mixing layer and this is discussed in terms of the particle history. The effect of gravity on the conditioned particle dynamics, in particular on the time of arrival statistics and radial distribution function, is addressed. Sponsored by the U.S. NSF.

1:52PM CV.00005 Inertial particles in a shearless mixing layer: direct numerical simulations, PETER IRELAND, LANCE COLLINS, Cornell University — Entrainment, the drawing in of external fluid by a turbulent flow, is present in nearly all turbulent processes, from exhaust plumes to oceanic thermoclines to cumulus clouds. While the entrainment of fluid and of passive scalars in turbulent flows has been studied extensively, comparatively little research has been undertaken on inertial particle entrainment. We explore entrainment of inertial particles in a shearless mixing layer across a turbulent-non-turbulent interface (TNI) and a turbulent-turbulent interface (TTI) through direct numerical simulation (DNS). Particles are initially placed on one side of the interface and are advanced in time in decaying turbulence. Our results show that the TTI is more efficient in mixing droplets than the TNI. We also find that without the influence of gravity, over the range of Stokes numbers present in cumulus clouds, particle concentration statistics are essentially independent of the dissipation scale Stokes number. The DNS data agrees with results from experiments performed in a wind tunnel with close parametric overlap. We anticipate that a better understanding of the role of gravity and turbulence in inertial particle entrainment will lead to improved cloud evolution predictions and more accurate climate models. Sponsored by the U.S. NSF.
therefore, a more complete picture of the dependency on Mach number. Simultaneous PIV/PLIF allows us to gather more complete measurements consisting of the time evolution of paired vorticity and density fields in the flow, and we will report statistical similarities and correlations between cloud spread and energy budgets of the particle phases. We then consider the gravitational settling of particle suspensions in triply periodic domains and determine the dependence of the settling rate on the concentration and the domain size. The numerical results are compared to well-known empirical data for settling in pipes, which is often used in continuum models for particle-laden flow.

**2:18PM CV.00007 Settling of finite-size colliding particles in unbounded domains**. JULIAN SIME-ONOV, Marine Geosciences Division, Naval Research Laboratory — A numerical model for Direct Numerical Simulations of particle-laden flows is developed to investigate the bulk behavior of particle suspensions. The particle hydrodynamic forces are determined by solving the incompressible Navier-Stokes equations for the finite Reynolds number flow around individual particles. At grid resolutions permitting large-scale simulations, the pressure and viscous stress are resolved everywhere except in the gap of colliding particles where micro-scale lubrication effects become important just before contact. An analytical expression for the unresolved pressure lubrication force is added as a correction to the numerically resolved hydrodynamic force. The mechanical-contact interaction between particles is modeled with Hookean elasticity and friction. The model predicted dissipation of particle momentum during collisions is compared to experimental data for the coefficient of restitution of immersed binary collisions. We then consider the gravitational settling of particle suspensions in triply periodic domains and determine the dependence of the settling rate on the concentration and the domain size. The numerical results are compared to well-known empirical data for settling in pipes, which is often used in continuum models for particle-laden flow.

**2:31PM CV.00008 Gravity influence on the clustering of charged particles in turbulence**. JIANG LU, HANSEN NORDSIEK, RAYMOND SHAW, Department of Physics, Michigan Technological University — We report results aimed at studying the interactions of bidisperse charged inertial particles in homogeneous, isotropic turbulence, under the influence of gravitational settling. We theoretically and experimentally investigate the impact of gravitational settling on particle clustering, which is quantified by the radial distribution function (RDF). The theory is based on a drift-diffusion (Fokker-Planck) model with gravitational settling appearing as a diffusive term depending on a dimensionless settling parameter. The experiments are carried out in a laboratory chamber with nearly homogeneous, isotropic turbulence, in which the flow is seeded with charged particles and digital holography used to obtain 3D particle positions and velocities. The derived radial distribution function for bidisperse settling charged particles is compared to the experimental RDFs.

*Research supported by the US National Science Foundation*

**2:44PM CV.00009 Dust SETTling in Protoplanetary Disks and the Onset of Kelvin-Helmholtz Instability**. JOSEPH BARRANCO, San Francisco State University, AARON LEE, EUGENE CHIANG, PHILIP MARCUS, University of California, Berkeley, XYLAR ASAY-DAVIS, Los Alamos National Laboratory — It is a remarkable fact that planets start out as microscopic grains within the protoplanetary disks of gas and dust in orbit around newly-formed protostars, somehow growing by a factor of \(10^{10}\) in mass in a period no more than \(10^7\) years. In the early stages of the planet formation process, small dust grains settle into the midplane of the disk in a few thousand years. As the dust layer gets thinner and denser, a vertical shear develops between the dust-rich layer at the midplane and the dust-poor gas above and below this layer. At great interest is under what conditions such a layer will be unstable to Kelvin-Helmholtz instability (KHI), which will remix the dust with the gas, thwarting the formation of planets. In our previous work, we worked in the single-fluid limit in which the local dust-to-gas ratio was an advectively conserved quantity (valid when the dust-gas friction time is very short). Here, we present new simulation in which this assumption is relaxed. We employ 2-fluid simulations of dust and gas to explore the evolution of a dust layer in the more general case in which the dust grains and gas can slip through each other. We will describe conditions that allow the dust layer to settle to sufficient density to gravitationally clump-up to form planetesimals before the onset of KHI.

**2:57PM CV.00010 Dispersion of a cloud of particles in the accelerated flow behind a moving shock**. GUSTAAS JACOBS, THOMAS DITTMANN, San Diego State University, WAI-SUN DON, Hong Kong Baptist University — We discuss the dynamics and dispersion of bronze particles that are initially arranged in varying cloud shapes and are accelerated in the supersonic flow behind a moving normal shock. Particle clouds with a particle volume concentration of 4% are arranged initially in a rectangular, triangular and circular shape, whose angle with respect to the incoming flow are also varied. Simulations are performed with a recently developed high-order resolution Eulerian-Lagrangian method, that approximates the Euler equations governing the gas dynamics with the improved high order weighted essentially non-oscillatory scheme, while individual particles are traced in the Lagrangian frame using high-order time integration schemes. The purpose of these simulations is two-fold: we are aiming to match a published shocktube experiment of the dispersion of an initially, nominally rectangular cloud shape behind a moving shock and we are aiming to validate our high-order methods against these experiments. The dynamics and resulting dispersion patterns of the developing particle-laden flows are distinctly different between different cloud shapes but we will report statistical similarities and correlations between cloud spread and energy budgets of the particle phases.

1The first author gratefully acknowledges support for this work from the AFOSR Young Investigator Program
1:13PM CW.00002 Effect of shear on mixing in R-T mixing layers at low Atwood numbers

BHANESH AKULA, Department of Mechanical Engg., Texas A&M University, MALCOLM ANDREWS, Las Alamos National Laboratory, DEVESH RANJAN, Department of Mechanical Engg., Texas A&M University — Effect of shear on R-T mixing is studied at two different Atwood numbers using the gas channel facility at Texas A&M University. The channel basically consists of two streams separated by a splitter plate. Pure air flows on top of the plate where as the lower density air Helium mixture flows on bottom and R-T mixing starts right after the splitter plate. Two different techniques, high resolution digital image analysis and simultaneous 3 wire cold wire Anemometry are used to measure R-T mixing growth rates. Results obtained from both the techniques are compared. Temperature is used as a marker to identify the streams and density is calculated from the temperature measured using a cold wire. Experiments are performed at Atwood numbers 0.04 and 0.1. At these Atwood numbers, effect of shear is studied by varying the velocity of one of the streams (mainly top stream). Simultaneous 3 wire cold wire Anemometry is performed at the vertical center line at three different axial locations. Different parameters obtained from these measurements including, $\theta$ (molecular mixing parameter), $\rho^{1/2}$ and vertical turbulent mass flux $\rho'v'$ and their effect on mixing growth rate are discussed.

1:26PM CW.00003 Miscible and immiscible liquid experiments and simulations on the Rayleigh-Taylor instability

MICHAEL ROBERTS, JEFFREY JACOBS, University of Arizona, WILLIAM CABOT, Lawrence Livermore National Laboratory — Experiments and numerical simulations are presented in which an incompressible system of two liquids is accelerated to produce the Rayleigh-Taylor instability. In these experiments, the initially stable, stratified liquid combination is accelerated downward in one of two experimental apparatus: a weight and pulley system in which a fluid filled container is accelerated on a rail system, or a new LIM apparatus which uses linear induction motors to accelerate the tank (which is attached to an aluminum plate) to produce much greater acceleration levels. Both miscible and immiscible liquid combinations are used. In both apparatuses the resulting fluid flows are visualized with backlight imaging using LED backlights in conjunction with monochrome high-speed video cameras, both of which travel with the moving fluid filled containers. Initial perturbations are either unforced and allowed to progress from background noise or forced by vertically oscillating the liquid combination to produce parametric internal waves. The results experiments are compared to numerical simulations performed using the CFD code Miranda.

1:39PM CW.00004 Experiments and theory on binary mixture evaporation in Hele-Shaw cells

JUTHAMAS KAMRAK, SAM DEHAECK, ALEXEY REDNIKOV, Service TIPs - Fluid Physics, ULB, B-1050 Brussels, Belgium, HSUEH CHING, FRÉDÉRIC DOUMENC, BEATRICE GUERRIER, Laboratoire FAST, UPMC, UPSUD, CNRS, Orsay F 91405, France, PIERRE COLINET, Service TIPs - Fluid Physics, ULB, B-1050 Brussels, Belgium — Evaporation of binary mixtures is studied using a Hele-Shaw cell. Refractive index variations during the evaporation are followed using a Mach-Zehnder interferometer. A Rayleigh-Taylor instability, due to the evaporation-induced density stratification, is observed during the process, both for simple and more complex mixtures. A theoretical model is developed for 1D concentration profiles before instability, in order to facilitate the interpretation of the experimental results. In the case of the aqueous solutions of ethanol, it takes into account evaporation of both the solute and the solvent. However, in the case of polymer solutions, only the solvent evaporates, and the properties of the solution (viscosity, diffusion coefficient, saturated vapor pressure, etc.) strongly depend on the time-dependent polymer concentration. The concentration profiles obtained from the theoretical model are compared to the experiments, for both systems.

1:52PM CW.00005 Experimental investigation of Rayleigh Taylor instability in elastic-plastic materials

AARON ALAN HALEY, ARINDAM BANERJEE, Missouri S&T — The interface of an elastic-plastic plate accelerated by a fluid of lower density is Rayleigh Taylor (RT) unstable, the growth being mitigated by the mechanical strength of the plate. The instability is observed when metal plates are accelerated by high explosives, in explosive welding, or in volcanic island formation due to the strength of the inner crust. In contrast to the classical case involving Newtonian fluids, RT instability in accelerated solids is not well understood. The difficulties for constructing a theory for the linear growth phase in solids is essentially due to the character of elastic-plastic constitutive properties which has a nonlinear dependence on the magnitude of the rate of deformation. Experimental investigation of the phenomena is difficult due to the exceedingly small time scales (in high energy density experiments) and large measurement uncertainties of material properties. We performed experiments on our Two-Wheeled facility to study the linear stage of the incompressible RT instability in elastic-plastic materials (yogurt) whose properties were well characterized. Rotation of the wheels imparted a constant centrifugal acceleration on the material interface that was cut with a small sinusoidal ripple. The controlled initial conditions and precise acceleration amplitudes are levered to investigate transition from elastic to plastic deformation and allow accurate and detailed measurements of flow properties.

2:05PM CW.00006 Transition to turbulence in shock-accelerated flows without reshock

M. LOMBARDINI, D.I. PULLIN, D.I. MEIRON, California Institute of Technology, R.A. GORE, Los Alamos National Laboratory — A numerical investigation of turbulence transition in shock-accelerated flow is described. Large-eddy simulations are performed for a heavy-light, SF$_6$-air ($A \approx -0.67$) perturbed, density impact interface by a shock wave of Mach number 1.5, 3.0 or 5.0. For these shock strengths, the initial perturbation amplitude is chosen such that the post-shock amplitude is about 25% of the initial perturbation dominant wavelength. The flow is computed in the frame of the unperturbed, post-shock interface and the LES uses periodic boundary conditions in the directions transverse to the main flow. This allows two isotropic directions within the mixing regime enabling calculation of instantaneous radial spectra. The spectra are obtained at the center-plane of the mixing zone at various times during the layer growth. Results indicate that the power spectra of the velocity components approach a $k^{-5/3}$ scaling, signaling a transition to turbulence accompanying a reorganization of the deposited kinetic energy. A spectral measure of the flow anisotropy shows a tendency to isotropy of the flow, although the axial-velocity power spectrum contains, at almost every scale, more than a third of the total kinetic energy. A budget of the plane-averaged, root-mean square vorticity accounts for the different sources of vorticity fluctuation and their evolution following the shock interaction.

2:18PM CW.00007 The Effect of Eccentricity on the Stability of Spiral Flows

PIETRO VALSECCHI, ExxonMobil Upstream Research Company — The instability mechanisms acting on the flow of fluid between two concentric cylinders where an axial pressure gradient is also present have been extensively studied and understood over the past three decades [1,2]. The eccentricity of one cylinder axis with respect to the other disrupts the axial symmetry that allows for the simplified analytical description of the base flow and introduces a radial component of the base flow. In the present study, a small eccentricity is introduced and the variation of the flow over the tangential direction is described by the first Fourier mode in $\theta$. With the origin at the center of the inner cylinder, the modifications to the governing equations are discussed, as well as the deviation in the boundary conditions given by the changing distance of the wall from the center. An analytical formulation of the base flow is derived and the bases for a linear stability calculation are laid.


2:31PM CW.00008 The effects of initial conditions on single and two-mode Rayleigh-Taylor instability, TIE WEI, DANIEL LIVESCU, MALCOLM ANDREWS, Los Alamos National Laboratory — The dependence on initial conditions of single and two-mode Rayleigh-Taylor instability (RTI) is investigated using Direct Numerical Simulations (DNS). A new stage, chaotic development, was found at very late time of single-mode RTI, after the re-acceleration stage. We found that details of the shape of the initial perturbation, such as the diffusion thickness and perturbation amplitude, have a strong effect on the growth rate during the early and late time development, but minimal during the potential flow regime, such that the Goncharov “terminal velocity” result remains robust. The early time evolution is sensitive to diffusive effects and the dependence on initial conditions can be minimized by increasing the Reynolds and/or Schmidt numbers. At very late time, single-mode RTI transitions into a chaotic development stage, with strong sensitivity to initial conditions. We have studied initial conditions on two-mode RTI, and found that the growth is strongly affected by the combination of mode numbers and amplitudes as well as the phase shift between modes. At late times, the motions become quite complicated, however some new phenomena, such as “leaning,” “ejection,” and “mode resonance,” can be identified as significantly influencing the growth rate.

2:44PM CW.00009 On Initial Conditions for Turbulent Rayleigh-Taylor Mixing, BERTRAND ROLLIN, MALCOLM J. ANDREWS, Los Alamos National Laboratory — Rayleigh-Taylor (RT) instability occurs when the pressure gradient opposes the density gradient at a perturbed interface between two media. For fluids, the instability causes mixing which, in time, turns turbulent. This fundamental instability is observed in nature phenomena such as salt dome formation, or supernovae explosions, and in engineering applications such as heat exchangers and sprays in internal combustor, or in the implosion phase of Inertial Confinement Fusion (ICF). Non negligible effects of initial conditions (ICs) on the development and turbulent mixing of the Rayleigh-Taylor instability, create an opportunity for prediction and “design” of RT turbulence for engineering purposes. Most turbulence models used for studying engineering applications are defined for fully developed turbulence, and therefore do not account for initial conditions effects. Our research seeks a rational methodology to provide initial conditions in variable density turbulence models. We report our methodology for following the evolution of the mixing layer based on the composition of the initial perturbation spectrum, and extracting profiles of relevant variables for the turbulence model. Metrics defining the time at which the turbulence model should relay our model, and when ICs-induced anomaly(s) will occur in late time turbulent mixing will also be discussed. Handling of late time anomaly in the turbulence model will be suggested.

2:57PM CW.00010 Effect of initial conditions on a high Schmidt-number Rayleigh-Taylor mixing layer, L.A.RAGHU MUTNURI, ARINDAM PANERJEE, Missouri S&T — An experimental investigation of the effect of initial conditions in spatiotemporal evolution of a high Schmidt number, low Atwood number (0.00075), reactive, turbulent Rayleigh-Taylor (RT) mixing layer will be presented. A horizontal solid barrier separating the participating fluids, contained in a static tank, is withdrawn to produce the RT unstable configuration and is similar in configuration to experiments of Dalziel (JFM-1999). The physical shape of the barrier coupled with the wake left on its withdrawal; define the spatial structure of initial perturbations. Passive control of initial conditions is attained by varying the shape of the barrier for a defined withdrawal rate. The design of the barrier is guided by integrated large-eddy simulations of the Euler equations in three dimensions with numerical dissipation, to the desired effect. Backlit imaging is used to study the temporal evolution of the RT mixing layer. Diffusion limited neutralization reaction in the presence of an indicator is used as a marker to quantify the extent of molecular mixing of the two fluids. The evolution of bubble growth coefficient, molecular mixing parameters and probability density functions are analyzed to draw comparison with existing numerical and experimental data.

Sunday, November 21, 2010 1:00PM - 3:10PM — Session CX Aerodynamics II Hyatt Regency Long Beach Regency D

1:00PM CX.00001 Flow over a Modern Ram-Air Parachute Canopy1, MOHAMMAD MOHAMMADI, HAMID JOHARI, California State University, Northridge — The flow field on the central section of a modern ram-air parachute canopy was examined numerically using a finite-volume flow solver coupled with the one equation Spalart-Allmaras turbulence model. Ram-air parachutes are used for guided airdrop applications, and the canopy resembles a wing with an open leading edge for inflation. The canopy surfaces were assumed to be impermeable and rigid. The flow field consisted of a vortex inside the leading edge opening which effectively closed off the canopy and diverted the flow around the leading edge. The flow experienced a rather bluff leading edge in contrast to the smooth leading of an airfoil, leading to a separation bubble on the lower lip of the canopy. The flow inside the canopy was stagnant beyond the halfway point. The section lift coefficient increased linearly with the angle of attack up to 8.5° and the lift curve slope was about 8% smaller than the baseline airfoil. The leading edge opening had a major effect on the drag prior to stall; the drag is at least twice the baseline airfoil drag. The minimum drag of the section occurs over the angle of attack range of 3° – 7°.

1Sponsored by the US Army Natick Soldier Systems Center.

1:13PM CX.00002 Effect of Leading and Trailing Edge Geometry on the Aeromechanic Characteristics of Membrane Aerioloifs1, SARA ARBOS, ZI PANG, BHARATHRAM GANAPATHISUBRAMANI, RAFAEL PALACIOS, Imperial College — The geometry of the rigid leading and trailing edges that hold the membrane could affect the aeromechanic performance of membrane wings. In this study the interaction between the supports and a membrane aerofoil is explored. Tests are performed at low Reynolds numbers, 4.3 × 104 - 1.1 × 105, and incidences of 0° - 30°. Four different leading and trailing edge geometries have been analysed focusing on the unsteady characteristics of the wake and the structural vibration of the membrane. Results indicate that aerelastic coupling between vortex shedding and membrane vibration depends upon the type of membrane support. The wake’s kinetic energy distribution has been found to be dependent on α for round supports but independent of α for rectangular supports. Finally, correlation between the membrane deflection and the lift generation has been found for all cases studied. Further analysis will be conducted and discussed.

1Funding from EPSRC through grant EP/F056206/1 is greatly appreciated.

1:26PM CX.00003 Characterization of a Three-Dimensional Turret Wake for Active Flow Control Part I: Simulation, CHRISTOPHER RUSCHER, PATRICK SHEA, RYAN WALLACE, JOHN DANNENHOFER, III, MARK GLAUSER, Syracuse University — The use of airborne optical devices has led to an increased need to study the flow around a turret. Separation around the turret causes density fluctuations that degrade the performance of the optical device. The separation region can be decreased using different methods of flow control, such as suction. A computational fluid dynamics code that employed Reynolds Averaged Navier-Stokes turbulence models was used to estimate the flow field around a turret and was compared with particle image velocimetry data for validation. The k-ω model performed better than the commonly used k-ε turbulence model when comparing the separation area and separation strength (integral of the negative streamwise component of velocity) on the center plane of a turret. The k-ε model predicted the separation area with an error of 74% and the k-ω model predicted the separation area with an error of 13%. Separation Strength was predicted with an error of 83% and 25% by the k-ε model and the k-ω model respectively. The more accurate k-ω model will be used to guide future flow control experiments.
1:39PM CX.00004 Characterization of a Three-Dimensional Turret Wake for Active Flow Control Part II: Experimental Study, PATRICK SHEA, CHRISTOPHER RUSCHER, RYAN WALLACE, MARK GLAUSE, JOHN DANNENHOFFER, III, Syracuse University — Experimental measurements have been performed to characterize the wake of a three-dimensional, non-conformal turret. Experiments were performed in a low-speed wind tunnel at Syracuse University using particle image velocimetry, hotwire anemometry and dynamic and static pressure measurements. The objective of the study was to characterize the spatial and temporal nature of the wake region as well as to investigate the importance of the incoming flow field. Computational studies have been performed in conjunction with this work to help guide the experimental study and offer insight into the complex three-dimensional flow field. With a better understanding of the wake and three-dimensional characteristics of the turret flow field, closed-loop, active flow control systems will be developed to help reduce fluctuating loading and aero-optical distortions associated with the turbulent flow field.

1:52PM CX.00005 Wind turbine performance predictions using propeller vortex lattice methods, BRENDEN EPPS, MIT, RICHARD KIMBALL, Maine Maritime Academy — A major concern in the design and operation of large wind turbines is unsteady blade loads. These can lead to fatigue failure, and hence are major structural design driver. Prediction and mitigation of peak blade loads is challenging, because the aerodynamic, structural, and controls problems are coupled, often non-linearly. An efficient computational tool is vital to analyze these types of problems early in the design cycle, before structural designs are frozen and full-blown CFD and FE analysis are performed. The authors present a computational tool for the design and analysis of large wind turbines. The numerical model is based on the propeller vortex lattice lifting line methods utilized by the US Navy as well as commercial propeller designers. The numerical model is implemented in an open-source code suite called OpenProp, which includes turbine optimization routines as well as performance analysis routines. Examples of turbine designs are presented, including actual parts and experimental performance data.

2:05PM CX.00006 Dynamic Surface and Flow-Field Measurements of a Pitching Wind Turbine Blade, JOHN STRIKE, MANJINDER SINGH, MICHAEL HIND, JONATHAN NAUGHTON, University of Wyoming — Dynamic pitching is used to study the unsteady aerodynamics of wind turbine blade airfoils. The dynamic flow field is characterized in a wind tunnel using surface pressure measurements coupled with Particle Image Velocimetry (PIV). To obtain the unsteady pressure distribution, a 10.16 cm chord DU97W-300 airfoil with 32 pressure ports has been coupled to a pitch transducer module through 1.07 m of 0.86 mm diameter tubing. Pressure data sampled at 500 Hz are used to estimate the unsteady surface pressure utilizing an optimal Wiener deconvolution method. PIV images are systematically acquired at different phases of the airfoil pitching cycle, and Proper Orthogonal Decomposition (POD) is used to reconstruct the unsteady flow field. To compare the current setup with previous studies that use the same airfoil geometry, pressure measurements are taken at a fixed angle of attack. The airfoils are then oscillated about mean angles of attack and amplitudes and frequencies up to 15 Hz that reflect the angle of attack range and reduced frequencies associated with wind turbines in the field. The combined measurements capture the links between flow-field structure and the observed surface pressures.

2:18PM CX.00007 A Free Wake Numerical Simulation for Darrieus Vertical Axis Wind Turbine Performance Prediction, RADIAN BELLU, Drexel University — In the last four decades, several aerodynamic prediction models have been formulated for the Darrieus wind turbine performances and characteristics. We can identify two families: stream-tube and vortex. The papers presents a simplified numerical techniques for simulating vertical axis wind turbine flow, based on the lifting line theory and a free vortex wake model, including dynamic stall effects for predicting the performances of a 3-D vertical axis wind turbine. A vortex model is used in which the wake is composed of trailing stream-wise and shedding span-wise vortices, whose strength are equal to the change in the bound vortex strength as required by the Helmholtz and Kelvin theorems. Performance parameters are computed by application of the Biot-Savart law along with the Kutta-Jukowski theorem and a semi-empirical stall model. We tested the developed model with an adaptation of the earlier multiple stream-tube performance prediction model for the Darrieus turbines. Predictions by our method are shown to compare favorably with existing experimental data and the outputs of other numerical models. The method can predict accurately the local and global performances of a vertical axis wind turbine, and can be used in the design and optimization of wind turbines for built environment applications.

2:31PM CX.00008 The aerodynamics of jumping rope, JEFFREY ARISTOFF, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We present the results of a combined theoretical and experimental investigation of the motion of a rotating string that is held at both ends (i.e. a jump rope). In particular, we determine how the surrounding fluid affects the shape of the string at high Reynolds numbers. We derive a pair of coupled non-linear differential equations that describe the shape, the numerical solution of which compares well with asymptotic approximations and experiments. Implications for successful skipping will be discussed, and a demonstration is possible.

2:44PM CX.00009 Numerical Investigations of a Wall Jet with Tabs, HAMID RAHAI, CARLOS ORRALA, HUY HOANG, CEERS/CSULB — Numerical investigations of a wall jet with tabs were performed. The tabs were rectangular thin metal plates placed at the jet outlet, either at the top mid boundary, or at the mid-sections of the three unbounded boundaries, perturbing into the jet. The analyses were carried out at a maximum mean velocity of 15 m/sec., which corresponds to an approximate jet Reynolds number based on the vertical jet dimension of 13027. The numerical calculations were performed using the Reynolds-Averaged Navier-Stokes equations with the Wilcox K-ω turbulence model. Results that include axial and spanwise variations of the mean velocity, velocity vector, turbulent kinetic energy and vorticity at different axial locations show that with a single tab, the spanwise entrainment is enhanced while with the three tabs, both the vertical and spanwise entrainments are increased. The increase in the spanwise entrainment should result in enhanced film cooling applications.

1Professor and Director
2Research Associate
3Lecturer

2:57PM CX.00010 On the skin friction drag reduction in large wind turbines using sharp V-grooved riblets. Application to a 2.5 MW Clipper wind turbine section, ROGER ARNDT, LEONARDO CHAMORRO, FOTIS SOTIROPOULOS, University of Minnesota — Skin friction drag reduction through the use of riblets has been a topic of intensive research during the last decades. Main efforts have been placed on both numerical (mainly DNS) and experimental approaches. In spite of the valuable efforts, the fundamental mechanisms that induce drag reduction are not well established. In this study, wind tunnel experiments were performed to quantify the drag reduction in a wind turbine airfoil using different V-groove riblet structures. A full-scale 2.5MW Clipper wind turbine airfoil section (of 1 meter chord length, typical of the 88% blade span), was placed in the freestream flow of the wind tunnel at the Saint Anthony Falls Laboratory, University of Minnesota. Four different sizes of V-groove riblets were tested at different angles of attack at full scale Reynolds number of Re=2.67×106 (based on the airfoil chord length). Force sensors were used to measure Lift and Drag. A combination of single and cross-wire anemometers were also used to study the turbulent scale-to-scale interaction in the near wall region to better understand the physical mechanisms of drag reduction and flow characteristics in that region. The measurements will be used to develop and test the performance of near-wall boundary conditions in the context of RANS and hybrid RANS/LES models.
flow correction and finite deformation of the interface. To study the evolution of the most amplifying disturbances in the two-fluid flow. Our methodology takes into account non-linear modal interactions, the mean flow distortion, nonlinear interactions, and finite amplitude effects to the development of the two-fluid structures. In addition to accuracy, the computational efficiency of our method is compared to direct Navier-Stokes simulations.

Non-linear evolution of vortical disturbances in two-fluid boundary layers.

These results may be of particular interest to researchers studying the transient growth and nonlinear stability of two-fluid flows. We also show that the “shear” viscosity ratio is larger than the thickness ratio of the two layers; these “interfacial” mode instabilities are also present when density stratification is destabilising.

Three-dimensional linear instability in pressure-driven two-layer channel flow of a Newtonian and a Herschel-Bulkley fluid.

The predictions of linear, locally-parallel analyses do not, however, take into account the spreading of the mean flow. The contribution of non-parallelism is quantified in the case of spatially developing boundary layers. Furthermore, beyond the early linear stage, the amplitude of the instability waves becomes appreciable and non-linear effects can no longer be neglected. As a result, an accurate description of the evolution of disturbances in two-fluid boundary layers must account for non-linear interactions and mean flow modification. We apply the framework of the non-linear Parabolized Stability Equations (PSE) in order to study the evolution of the most amplifying disturbances in the two-fluid flow. Our methodology takes into account non-linear modal interactions, the mean flow correction and finite deformation of the interface.

Three-dimensional spatio-temporal instabilities in two-layer flows at high Reynolds numbers.

The predictions of linear, locally-parallel analyses do not, however, take into account the spreading of the mean flow. The contribution of non-parallelism is quantified in the case of spatially developing boundary layers. Furthermore, beyond the early linear stage, the amplitude of the instability waves becomes appreciable and non-linear effects can no longer be neglected. As a result, an accurate description of the evolution of disturbances in two-fluid boundary layers must account for non-linear interactions and mean flow modification. We apply the framework of the non-linear Parabolized Stability Equations (PSE) in order to study the evolution of the most amplifying disturbances in the two-fluid flow. Our methodology takes into account non-linear modal interactions, the mean flow correction and finite deformation of the interface.

Numerical simulations of two-fluid channel flow with wall deposition and ageing effects.

We study the dynamics of two immiscible fluids with a high viscosity contrast in pressure-driven channel flow using direct numerical simulations at moderate Reynolds numbers. The equations of mass, momentum and energy conservation in both fluids are solved using a procedure based on the diffuse interface method. The simulations study the effect of waves, generated by a random 3D noise, at the inlet on the spatio-temporal behaviour of the instabilities. Of specific interest are the conditions of growth/death of the spanwise interfacial perturbation. Preliminary results show a sustained growth of the spanwise mode, irrespective of the primary streamwise mode, at various streamwise locations in the domain. At positions close to the inlet the spanwise wave grows linearly until a non-linear distortion which eventually saturates the amplitude. This work extends our recently reported two-dimensional studies on spatiotemporal interfacial instabilities (J. Fluid Mech. (2010), vol. 656, pp. 458–480) to i) three dimensions and ii) higher Reynolds numbers.

Dynamics of surfactant-laden lenses.

We consider the dynamics of lenses of immiscible fluids laden with surfactant. We use lubrication theory to derive equations for the positions of the interfaces and the surfactant concentrations. The surfactant is allowed to exist in the form of monomers as well as micelles and the model accounts for the effects of surfactant on the moving contact line. We use a finite-element formulation to obtain numerical solutions of the evolution equations and carry out a full parametric study. Our results catalogue the various types of behaviour encountered, which range from complete spreading of the lens, to spreading followed by retraction, to sustained oscillations.
2:18PM CY.00007 Surfactant-induced superspreading of liquid drops on solid substrates1. GEORGE KARAPETAS, RICHARD CRASTER, OMAR MATAR, Imperial College London — The mechanisms driving surfactant-enhanced superspreading of droplets on solid substrates are investigated. Lubrication theory for the droplet motion, and advection-diffusion equations as well as chemical kinetic fluxes for the surfactant transport, lead to coupled evolution equations for the drop thickness, interfacial concentrations of surfactant monomers and bulk concentrations of monomers and micellar aggregates. The surfactant is allowed to adsorb on the substrate either directly from the bulk monomer concentrations or from the liquid-air interface through the contact line. The evolution equations are solved numerically using the finite element method. The results show that basal adsorption of surfactant plays a crucial role in the spreading process: the continuous removal of surfactant from the liquid-air interface, due to the adsorption at the solid surface, is capable of inducing high Marangoni stresses, close to the droplet edge, driving very fast spreading. In many of the cases studied, the droplet radius, $R$, grows with time, $t$, with power laws of unity or even higher, close to the reported experimental values for superspreading. The spreading rates depend non-monotonically on the initial surfactant concentration also in accordance with experimental observations. In certain cases, the spreading droplet forms a rim at its leading edge, or creates a “secondary” front separated from its main body.

1EPSRC Grant number EP/E056466

2:31PM CY.00008 Dynamics of surfactants spreading on gel layers: cracking and pattern formation1. CONSTANTINE SPANDAGOS, SHERMIN AKHTAR, PAUL LUCKHAM, OMAR MATAR, Imperial College London — The spreading of surfactant droplets on gel layers is observed to be accompanied by “starbursts” resembling cracking patterns on the gel surface. Marangoni stresses induced by surface tension gradients between the spreading surfactant and the underlying gel layer are identified to be the driving force behind these phenomena. A parametric study that involves different surfactants on various gels aims to investigate the ways that system parameters such as the surfactant chemistry and concentration and the gel type and strength can affect the morphology and dynamics of the cracking patterns. The surfactants used in this study include SDS (Sodium Dodecyl Sulphate) and the “super-spreader” Silwet L-77 (a Trislopane ethoxylate); the different gel substrates are made of agar and gelatine. The instability associated with the cracking on the surface of the gels is characterised in terms of the number of “arms” that forms and of their mean length as a function of time. Qualitative information on the surfactant distribution and gel topography, where the patterns are formed, is also obtained.

1EPSRC Grant number EP/E056466

2:44PM CY.00009 Gravitational stabilization of the interfacial surfactant-induced instability of shear flows. ADAM J. SCHWEIGER, ALEXANDER L. FRENKEL, DAVID HALPERN, University of Alabama — The linear stability of a two-layer plane Couette-Poiseuille flow with an insoluble surfactant on the interface in the presence of gravity is considered. Previous work has shown that when gravity is absent, the interfacial surfactant in the incompressible inertialess shear flow implies its instability. Considering now the case when gravity is included and the denser fluid is at the bottom, only the normal modes whose wavenumbers $\alpha$ are smaller than some marginal value $\alpha_0$ are expected to be unstable. Also, $\alpha_0$ should decrease as the Bond number $Bo$ (proportional to the acceleration of gravity) increases. A natural question is, as $Bo$ increases, does $\alpha_0 \rightarrow 0$ as $Bo \rightarrow Bo_0$, some finite threshold value? The answer is “no” for both the infinite and finite thickness ratios, but in differing ways. By the standard normal mode approach, the dispersion equations found to be quadratic in $\gamma$, the complex “growth rate.” It yields the dispersion relation $Re_{\gamma} = Re(\alpha; Bo, M, s, m)$, where $M$ is the Marangoni number, $m$ is the viscosity ratio, and $s$ is the bottom-side interfacial shear rate. The theory goes without the lubrication approximation: it accounts for the normal modes of all wavelengths.

2:57PM CY.00010 Long-wavelength Marangoni convection in liquid layer with insoluble surfactant in modulated thermal field. ALEXANDER MIKISHEV, ALEXANDER NEPOMNYASHCHY, Technion — Marangoni convection in a horizontal liquid layer with an insoluble surfactant distributed on the flat nondeformable free surface is considered. The temperature flux applied to the rigid bottom boundary of the horizontal layer is modulated periodically near a fixed mean value. On the free surface the Biot number is assumed to be small (poorly conducting surface). The surface tension varies linearly with temperature and surface concentration. It has been found formerly that in the absence of temperature flux modulations there exist two longwave instability modes, monotonic and oscillatory ones, which determine the instability threshold. The linear analysis shows that in the case of the monotonic mode, the periodic modulation of the heat flux increase the critical Marangoni number. This effect is especially strong in the region of a small modulation frequency. In the case of the oscillatory mode the influence of modulation depends on the parameters of the problem. A weakly nonlinear analysis near the stability threshold is performed. The research was supported by the grant #3-5799 of the Israeli Ministry of Science and partially supported by the EU via the FP7 Marie Curie scheme (grant #PIIT-GA-2008-214919 (MULTIFLOW)).

Sunday, November 21, 2010 1:00PM - 2:57PM – Session CZ Waves II Hyatt Regency Long Beach Regency F

1:00PM CZ.00001 Rotating Solitary Wave during Liquid Drainage. MOHAMED FAYED, Concordia University, HAMID AIT ABDERRAHMANE, McGill University, GEORGIOS H. VATISTAS, HOI DICK NG, Concordia University — This work reports on the observation of a rotating solitary wave during liquid drainage from a cylindrical reservoir when shallow water conditions are reached. Using results obtained from a high-speed camera we manage to observe for the first time the formation of a rotating solitary wave on the liquid-air interface. The theory goes without the lubrication approximation: it accounts for the normal modes of all wavelengths.

1:13PM CZ.00002 Can non-propagating hydrodynamic solitons be forced to move? LEONARDO GORDILLO, MARCEL G. CLERC, NICOLAS MUJICA, TANJA SAUMA, YAIR ZARATE, Universidad de Chile, IGNACIO ESPINOZA, Pontificia Universidad Catolica de Chile — Development of technologies based on localized states depends on our ability to manipulate and control these nonlinear structures. In order to achieve this, the interactions between localized states and control tools should be well modelled and understood. We present a theoretical and experimental study for handling non-propagating hydrodynamic solitons in a vertically driven rectangular water basin, based on the inclination of the system. Experiments have shown that tilting the basin induces non-propagating solitons to drift towards an equilibrium position through a relaxation process. Our theoretical approach is derived from a family of generalized nonlinear Schrödinger equation which models the system. The basin tilting effect is incorporated as a spatially inhomogeneous linear correction on dissipation. A motion law for hydrodynamic solitons can be deduced from these assumptions. The model equation, which includes a constant speed and a linear relaxation term, nicely reproduces the motion observed in our experiment.

1:26PM CZ.00003 Stationary solutions of the extended reduced Ostrovsky equation. MARIA OBREGON, Universidad de Malaga (Spain), YURY STEPANYANTS, University of Southern Queensland (Australia), RAMON FERNANDEZ-FERIA, Universidad de Malaga (Spain) — The extended Ostrovsky equation describes large-amplitude internal oceanic waves affected by Earth’s rotation, including an additional cubic term to take into account the effect of strong nonlinearities. Its reduced version, in which the small-scale, or Boussinesq, dispersion term is omitted, is relevant for the description of long internal waves in oceans, when their wavelengths are much larger than the basin depth. It may be of interest also for waves of other physical origin in nonlinear media with large-scale dispersion. In this work we present a systematic analysis of the stationary solutions to this extended reduced Ostrovsky equation and their categorization. Periodic and solitary solutions are constructed and their typical parameters are estimated for the natural oceanic conditions.
1:39PM CZ.00004 Selection Rules for Internal Gravity Waves and Inertial Waves, CHUNG-HSIANG JIANG, PHILIP MARCUS, University of California, Berkeley — Perturbation methods are used to calculate nonlinear interaction of waves, however most analyses skip the question as to whether the \( n \)th order solutions exist. The dispersion relation for internal gravity waves does not relate the magnitude of the wave vector and its frequency, rather it relates the frequency and direction of the wave vector. Thus, spatially collocated beams of internal waves are made of a continuum of plane waves with different wavenumbers, but the same frequency. For two parent beams to create a daughter, the plane waves within the parent and daughter beams must obey the triad condition (the spatial wave vector of the daughter equals the sum of the parents’ vectors, and temporal frequency of the daughter equals the sum or difference of the parents’ frequencies) and the dispersion relationship. Contrary to what is assumed implicitly, these conditions cannot always be satisfied. If they could, then the interaction of two beams of gravity waves would produce 8 daughter beams, consisting of (each with 4 beams). The beams in one cross have a frequency equal to the sum of the frequencies of the parents and the beams in the other have a frequency equal to the difference. Most of these daughter beams cannot exist. We derive selection rules for the beams. We extend our analysis to a more generic set of waves.

1:52PM CZ.00005 The Combined Effects of Light-wind and Surfactants on Spilling Breakers, J.H. DUNCAN, X. LIU, D. WANG, University of Maryland — Spilling breaking waves in the presence of light-winds and surfactants were studied experimentally in a wind-wave tank. The breaking waves were mechanically generated with a single wave maker motion that produces a weak spilling breaker in clean water without wind. Separate experiments were performed with the same wave maker motion and very low wind speeds in clean water and in water with various concentrations of Triton X-100 (a soluble surfactant). The crest-profiles of the waves along the center plane of the tank were measured with a cinematic laser-induced fluorescence technique. In clean water with a wind speed lower than 2.3 m/s (the minimum wind speed of wind-generated waves in our tank), the wave breaking is initiated with a bulge-capillary-ripple pattern. When the wind speed is above 2.3 m/s, wind waves are generated. These wind waves steepen on the front face of the crest of the mechanically generated waves and trigger breaking of these larger scale waves. In the presence of surfactants, the bulge-capillary-ripple pattern occurs at even higher wind speeds (3 m/s). Geometrical parameters describing the wave crest shape were found to scale with the wind speed to the third power.

The support of the National Science Foundation under grant OCE751853 is gratefully acknowledged.

2:05PM CZ.00006 Laboratory Measurements of Droplets Generated by Breaking Water Waves, J.H. DUNCAN, X. LIU, D. WANG, University of Maryland — The production of droplets generated by breaking water waves greatly affects the heat, mass and momentum transfer between the atmosphere and the sea surface. In this study, the generation of droplets by single breaking water waves, was explored in a wave tank. Plunging breakers were generated from dispersively focused wave packets (average frequency 1.15 Hz) using a programmable wave maker. The profile histories of the breaking wave crests along the center plane of the tank were measured with a cinematic laser-induced fluorescence technique, while the droplet diameters and motions were measured with a double-pulsed cinematic shadowgraph technique. The two measurement systems were mounted on an instrument carriage that was set to move along the tank following the breaking crests. It was found that droplets are primarily generated when the wave’s plunging jet generates strong turbulence during impact with the wave’s front face and when large air bubbles, entrapped during the plunging process, rise to the free surface and pop.

This work is supported by the National Science Foundation, Division of Ocean Sciences under grant OCE751853.

2:18PM CZ.00007 Effects of Rotation on Internal Solitary Waves, KARL HELFRICH, Woods Hole Oceanographic Institution, ROGER GRIMSHAW, Loughborough University, TED JOHNSON, University College London — An internal solitary waves in a rotating system decays by radiation into longer Poincare waves. The radiation extinguishes the initial solitary wave in a finite time. Recent numerical and theoretical studies show that the radiated waves develop into a localized nonlinear internal wave packet, or envelope soliton, that persists for long times. The 13-meter LEGI-Coriolis platform in Grenoble, France was used to perform laboratory experiments designed to test these theoretical results. The experiments confirm theoretical predictions of the packet formation and characteristics including the phase speed of the carrier waves and packet group speed. In particular, the wave number of the carrier wave is found to be close to the linear wave with the maximum group velocity. The localized packet formation is, however, a consequence of nonlinearity. As the rotation rate is increased, the initial disturbance (produced from a dam-break) develops into the packet structure without first forming a distinct solitary wave. These nonlinear inertia-gravity wave packets are a natural outcome in a rotating system and do not require initiation by radiation decay of an internal solitary wave.

2:31PM CZ.00008 Self-induced shear instabilities by large amplitude internal waves, CLAUDIO VIOTTI, University of North Carolina, ROBERTO CAMASSA, ROXANA TIRON, University of Southern California, UNC NSF CMG WAVE GROUP COLLABORATION — Large amplitude internal waves in sharply stratified fluids can generate large shear flows supporting Kelvin-Helmholtz instabilities. This talk will present a combined theoretical and numerical study of this instability. Spectral analysis of the corresponding non-self adjoint Taylor-Goldstein type equation will be outlined and illustrated on a specific model of internal wave structure. Subsequently, we will focus upon the asymmetric intensification of the instability by weakly non-parallel shear perturbations along long wave shear profiles.

2:44PM CZ.00009 The benthic layer under fully-nonlinear internal solitary waves of depression, YUNCHENG LIN, LARRY REDEKOPP, University of Southern California — Long internal waves are common features on the continental shelf and in lakes, but their dissipation via benthic boundary layer drag is largely unknown, particularly when the wave amplitudes are large and boundary layer corrections based on linear theory are clearly invalid. In general, the wave-induced boundary layer experiences a continuous favorable-to-adverse variation of the pressure gradient, undergoes transition, may reach a strongly turbulent state, and frequently separates near the point of maximum adverse pressure gradient in the lee of the wave. In this study a model for fully-nonlinear solitary waves of depression in a two-layer stratification is employed as the inviscid base state, and a RANS solver with \( \omega \) turbulence model is used to compute the stationary boundary layer under the wave. Local friction coefficients and eddy viscosities are computed in the footprint of the wave. Locations of boundary layer separation are computed as well as the integrated frictional drag over the region of attached boundary layer flow. Boundary layer of two beams of gravity waves ranging on shelf conditions, containing numbers, and surface roughness in an attempt to provide a quantitative measure of the frictional drag of long internal waves in realistic, shallow environs.

Sunday, November 21, 2010 3:30PM - 4:05PM — Session DS Invited Session: Particle Laden Viscous Fluids and the Gulf of Mexico Oilspill, Long Beach Convention Center Grand Ballroom A
3:30PM DS.00001 Particle Laden Viscous Fluids and the Gulf of Mexico Oilspill \footnote{The research is supported by NSF grant DMS-1048840 and a UC Lab Fees Research Grant.} \textsc{Andrea Bertozzi}, UCLA — The dynamics of particulates (e.g. sand) in viscous fluids is a complex process involving hindered settling dynamics and dynamics of the fluid such as shear. There is a renewed interest in understanding the dynamics of viscous fluids interacting with particulates due to the large volume of crude oil washing up on beaches in the Gulf of Mexico. I will address both modeling and mathematical challenges associated with this problem. Settling of particles in particle-oil mixtures on an incline can be quantitatively analyzed by a bifurcation that occurs between regimes of particle settling downstream of the flow and clear fluid separating out from the flow. An equilibrium theory compares shear-induced migration due to the bulk flow properties with hindered settling due to gravity, and matches well with laboratory experiments. I discuss current work on dynamic models for such particle-fluid mixtures and possible future work directed at understanding the formation of different classes of oil deposits on beaches in the Gulf.

Sunday, November 21, 2010 3:30PM - 4:05PM –
Session DT Invited Session: Fluid Simulation in the Movies: Navier and Stokes Must Be Circulating in Their Graves \textsc{Jerry Tessendorf}, Rhythm and Hues Studios — Fluid simulations based on the Incompressible Navier-Stokes equations are commonplace computer graphics tools in the visual effects industry. These simulations mostly come from custom C++ code written by the visual effects companies. Their significant impact in films was recognized in 2008 with Academy Awards to four visual effects companies for their technical achievement. However artists are not fluid dynamists, and fluid dynamics simulations are expensive to use in a deadline-driven production environment. As a result, the simulation algorithms are modified to limit the computational resources, adapt them to production workflow, and to respect the client’s vision of the film plot. Eulerian solvers on fixed rectangular grids use a mix of momentum solvers, including Semi-Lagrangian, FLIP, and QUICK. Incompressibility is enforced with FFT, Conjugate Gradient, and Multigrid methods. For liquids, a levelset field tracks the free surface. Smooth Particle Hydrodynamics is also used, and is part of a hybrid Eulerian-SPH liquid simulator. Artists use all of them in a mix and match fashion to control the appearance of the simulation. Specially designed forces and boundary conditions control the flow. The simulation can be an input to artistically driven procedural particle simulations that enhance the flow with more detail and drama. Post-simulation processing increases the visual detail beyond the grid resolution. Ultimately, iterative simulation methods that fit naturally in the production workflow are extremely desirable but not yet successful. Results from some efforts for iterative methods are shown, and other approaches motivated by the history of production are proposed.

Sunday, November 21, 2010 4:10PM - 6:20PM –
Session EA Mini-Symposium on Optical Effects of Turbulence \textsc{Edward Fry}, Texas A&M University-Corpus Christi — The turbulent flow impresses a subtle modification on the passing light and they are most pronounced in the nearforward direction i.e. in the direction of the light propagation. These effects can be observed either in phase (via the arrival angle) measurement or they can be observed in polarimetric measurements, as the change of light polarization state. Our measurements were carried out in a Rayleigh-Bernard convective tank with dimensions 0.3 m × 0.3 m × 0.3 m and flow with Rayleigh numbers 10^8 to 5 · 10^9. Using profiling thermistor and collocated 2D PIV system we have derived spatial temperature spectra collocated with the flow velocity fields. The length of the time series spans a few large eddy times, allowing the capture of energy and temperature fluctuations. The tank was also equipped with optical measurements which included 1D Shack-Hartmann wavefront sensing system and CCD based diffractometer allowing us to measure the nearly instantaneous respectively temperature spectra and the light beam linear polarization state. The optically obtained 3D temperature spectra were in agreement with fast thermistor spatial spectra. The optically measured temperature spectra in the far dissipation region yield Kraichnan scalar spectrum as predicted by numerical simulations. The diffractometer results show that the turbulence-induced depolarization rate depends on the strength of the turbulent flow, suggesting that light beam depolarization from turbulent flow may contain useful information regarding the smallest length scales of turbulent flow.

4:36PM EA.00002 Remote Sensing of Sound Speed in the Ocean via Brillouin Scattering \textsc{Edward Fry}, Texas A&M University — Innovative Brillouin LIDAR concepts to obtain range-resolved remote measurements of sound speed (and temperature) in the ocean are described. Objectives are an accuracy of 0.2 m/s (0.1degC) over a range of ~ 100 m in clear ocean with a range resolution of 1 m. Our approach provides high-resolution spectroscopic capabilities even in very severe acoustic/vibration environments. The detection is based on the use of edge filters to provide a high-resolution determination of the Brillouin frequency shifts. Edge filters could be molecular iodine absorption lines or excited-state Faraday anomalous dispersion optical filters. The transmitter is a commercial, injection seeded, frequency-doubled Nd:YAG laser that we have modified in two ways. First, we changed its operating temperature to obtain lasing at a frequency consistent with our choice of iodine absorption lines. Second, we implemented the Ramp and Fire technique we had developed so that the laser operates in a single longitudinal mode even when there are severe environmental disturbances. Test results will be presented that clearly demonstrate the efficacy of this new concept.

5:02PM EA.00003 An Overview of Aero-Optics \textsc{E.J. Jumper}, University of Notre Dame — Although Aero-Optics deals with the aberrating effects of variable index-of-refraction turbulent flows on lasers projected through them, it is specifically limited to compact turbulence due to flow over exit or receiving apertures, as opposed extended propagation paths through the atmosphere, for example. The origin of the word “Aero” in the description of the discipline is because compact, aberrating, turbulent flows are concomitant to airborne laser/optical systems. This talk will give a general overview of aero-optics starting with work in the 1970’s and progressing through its resurgence in the 1990’s due to the emphasis on shorter-wavelength lasers. The maturation of our understanding of the causes of the aberrating characteristics of aero-optical turbulence will also be discussed. Included in the talk will be the advances in wavefront-sensing technology and how this has led to the ability to properly scale aero-optic data and develop models for predicting its affects on system performance and explore mitigation techniques. The continued advance of these instruments is now allowing us to get a better appreciation for the impact of aero-optic flows on free-space communication even when the flow remains attached over the aperture, a condition that has little impact on directed energy systems.
5:28PM EA.00004 Remote Sensing of Turbulence in Natural Fluids

CARL H. GIBSON, University of California at San Diego — It is generally agreed that natural fluids including the atmosphere, ocean, and astrophysical objects are mixed by turbulence against the forces of gravity. However, the basic mechanisms, definition, and even the direction of the turbulence kinetic energy cascade remain controversial. Broadband remote sensing gives strong evidence to resolve such questions. Turbulence is found to be an eddy-like state of fluid motion where the inertial-vortex forces of the eddies are larger than any other forces that tend to damp the eddies out. Irotational flows are non-turbulent by definition. Because turbulent vorticity is always produced at the Kolmogorov scale, the direction of the turbulent energy cascade is always from small scales to large. Fossilization of the turbulence occurs at its largest scales. Fossil turbulence is any perturbation of a hydrophysical field produced by turbulence that is no longer turbulent at the scale of the perturbation. In the ocean, fossil vorticity turbulence internal waves carry bottom turbulence energy to the sea surface by means of beamed zombie turbulence maser action mixing chimneys, a generic process of natural fluids. Spectral analysis of the sea surface brightness from space satellites combined with simultaneous ocean microstructure sea truth reveals the generic mechanism, also supported by recent astrophysical observations http://arxiv.org/abs/1005.2772v4.

1 Depts. of MAE and SIO, CASS, La Jolla, CA 92039-0411

5:54PM EA.00005 Optical distortions by compressible turbulence: distortion measures and importance of small-scale structures

ALI MANI, Stanford University — This study is focused on aero-optics, which investigates optical distortions due to the compressibility mechanism, rather than entropy fluctuations. In particular, distortion effects by separated shear layers and turbulent wakes are considered. Typically, wavefront aberrations by these flows are larger than the optical wavelength; therefore, traditional measures (i.e. the Strehl ratio) would be inaccurate if used for quantifying optical distortions. Through statistical analysis of highly aberrated waves we introduce alternative measures with provable scaling properties. These norms, provide explicit relations between far-field optical statistics and statistics of the distorting media. We also present results of our study on the optical importance of small-scale flow structures. Using Kolmogorov hypothesis, a relation is derived to estimate the smallest optically-important length scale in a general aero-optical framework. This length is typically in the inertial range and the developed criterion is shown to reasonably predict the resolution requirements for simulations. This analysis can also be used to estimate frequency requirements for adaptive-optics.

Sunday, November 21, 2010 4:10PM - 6:20PM –
Session EB Turbulent Boundary Layers III Long Beach Convention Center 101B

4:10PM EB.00001 Understanding and quantifying wall-turbulence: a new closure approach

ZHEN-SU SHE, XI CHEN, YOU WU, Peking Univ., FAZLE HUSSAIN, Univ. of Houston — A new closure approach - structure ensemble dynamics (SED) - is proposed for integrating the flow dynamics into a rigorous, quantitative description of the mean flow of wall-bounded turbulent. Starting with the ensemble-averaged Navier-Stokes (EANS) equations, it expresses the unknown effects of fluctuation structures in terms of a set of order functions (a concept in statistical physics describing transitions between different statistical states). A multi-layer picture of wall turbulence naturally arises as a formal, quantitative extension of traditional views in terms of sublayer, buffer layer, log layer and wake. The order functions capture transitions between the layers. SED theory, applied to turbulent channel flow, reveals a surprisingly simple structure of $1 - z^4$ (where $z$ is the distance from the channel center) for a turbulent eddy length function $L_z$ in the bulk flow of the channel, where the traditional logarithmic layer near the wall is already included (without any matching): $L_z \propto y^{1/2}$ as $z \to 1$. A quantitative multi-layer model for relevant order functions is shown to give accurate description of the mean quantities like the profiles of mean velocity, Reynolds stress, kinetic energy, turbulence production and energy dissipation. Finally, a systematic procedure for evaluating numerical simulations using the SED theory is outlined.

4:23PM EB.00002 Scaling and interactions of inner and outer regions in wall-bounded turbulence

ROMAIN MATHIS, NICHOLAS HUTCHINS, IVAN MARUSIC, University of Melbourne, UNIVERSITY OF MELBOURNE TEAM — Recent investigations in wall-bounded turbulent flows have shown a nonlinear scale interaction, whereby the large-scale motion amplitude modulates the small-scale structures (Mathis et al., J. Fluid Mech. 628, 2009). Here, we present a comparison of the amplitude modulation effects between channels/pipes and boundary layers. It is found, that despite different large-scale structures in these internal and external wall-bounded flows, the amplitude modulation effects remain invariant in the inner region, whereas subtle differences appear in the outer region. Further details will be given on the close relationship between the wall-normal evolution of the amplitude modulation coefficient and the skewness as recently pointed out by Schlatter and Orlu (Phys. Fluids 22, 2010). Scaling issues related to the inner and outer regions will also be considered by using a scale-decomposition approach.

4:36PM EB.00003 Time-resolved evolution of the wall-bounded vorticity cascade

ADRIÁN LOZANO-DURÁN, U. Politécnica Madrid, JAVIER JIMÉNEZ, UPM and CTR Stanford — We study the temporal evolution of vortex clusters in turbulent channels with $Re_T = 950$ and 1880, using DNS sequences with temporal separations among fields short enough for individual structures to be tracked. From the geometric intersection of structures in consecutive fields, we build temporal connections of graphs of all the cluster interactions, and, from their properties, distinguish the “trunk” of each evolution from less important “branches.” It is found that the lifetimes of the connected families of attached clusters are proportional to the cube roots of their maximum volumes, of the order of $Tu_x/h = 0.25$ for the largest ones, and that they move approximately with the overall advection velocity. Special attention is paid to the origin of the attached structures, and to their relation with an inverse cascade. They tend to be born below $y^+ = 100$, and to grow upward, although a similar study of the Reynolds stresses suggests interactions in both directions. Merging of comparable clusters is common, but splitting tends to involve smaller fragments. The creation and evolution of new clusters during the bursting events of the logarithmic layer are also studied.

1 Funded CICYT

4:49PM EB.00004 Direct simulation of the zero-pressure-gradient boundary layer up to $Re_{\theta} = 6000$

JUAN A. SILLERO, GUILLÉM BORRELL, AYSE G. GUNGOR, JAVIER JIMÉNEZ, U. Politécnica Madrid, ROBERT D. MOSER, TODD A. OLIVER, U. Texas Austin — Preliminary results are presented from a direct simulation of the zero-pressure-gradient turbulent boundary layer in the range $Re_{\theta} = 2500-6000$, approximately matching channels at $Re_{\theta} = 2000$. Special emphasis is put on the effect of enforcing inflow conditions at a relatively-high Reynolds number, and on their influence on the streamline development of the mean and fluctuating flow properties.

1 Funded CICYT, INCITE
from the developed turbulent region. We are attempting to perform this data analysis even further upstream in the transitioning flow at $Re_{θ}$. This course, much larger than that obtained by including both turbulent and non-turbulent information there, and is consistent with a value obtained by extrapolating higher Reynolds number, especially in the buffer and sublayers. The skin friction coefficient, determined in this conditional manner in the transitional flow is, of course, much larger than that obtained by including both turbulent and non-turbulent information there, and is consistent with a value obtained by extrapolating from the developed turbulent region. We are attempting to perform this data analysis even further upstream in the transitioning flow at $Re_{θ} = 300$ where the turbulent spots are individuated. These results add further evidence to support the view that the structure of a developed turbulent boundary layer is different from its structure in its embryonic form in turbulent spots. *CTR 2010 Summer Program research.

Hairpin vortices in the transitional and developed turbulence of a flat-plate boundary layer*, J. WALLACE, Univ. of Maryland, X. WU, Royal Military College of Canada, I. PARK, P. MOIN, Stanford Univ. — The use of vortex lines to reveal vortical structures in turbulent shear flows has been in disfavor for some time, in spite of their successful use by Kim and Moin (1986, JFM 162) and Rogers and Moin (1987, JFM 176). This is because they are field lines that can be drawn wherever the flow is rotational, regardless of whether a true vortex exists in a part of the field or not. For this reason, it would be better to call them vorticity lines rather than vortex lines. A virtue that such lines have, however, is that the vortical structures they can reveal do not depend on setting a detection threshold, unlike all the vortex identifiers based on the velocity gradient tensor or based on a low pressure criterion. Furthermore, vorticity lines can be used to isolate a single vortical structure. We will show that individual hairpin vortices can be identified using vorticity lines in the transition region at $Re_{θ} = 500$, where turbulent spots merge, and in the developed turbulence at $Re_{θ} = 1850$ within a recent DNS of a flat-plate boundary layer, and that the vortices so identified have quite similar characteristics. These vortices emerge out of sheets of unorganized vorticity in the viscous sublayer. An attempt will be made to follow the temporal and spatial evolution of these vortical structures using simulation files closely separated in time. *CTR 2010 Summer Program research.

Transitional and turbulent flat-plate boundary layers with heat transfer . XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Center for Turbulence Research, Stanford University — We report on our direct numerical simulation of two incompressible, nominally zero-pressure-gradient flat-plate boundary layers from momentum thickness Reynolds number 80 to 1950. Heat transfer between the constant-temperature solid surface and the free-stream is also simulated with molecular Prandtl number $Pr = 1$. Throughout the entire flat-plate, the ratio of Stanton number and skin-friction $ST/C_f$ deviates from the exact Reynolds analogy value of 0.5 by less than 1.5%. Turbulent Prandtl number $P_r$ peaks at the walls. Preponderance of hairpin vortices is observed in both the transitional and turbulent regions of the boundary layers. In particular, the internal structure of merged turbulent spots is hairpin forest; the internal structure of infant turbulent spots is hairpin packet. Numerous hairpin vortices are readily detected in both the near-wall and outer regions of the boundary layers up to momentum thickness Reynolds number 150. This suggests that the hairpin vortices in the turbulent region are not simply the aged hairpin forests convected from the upstream transitional region. Temperature iso-surfaces in the companion thermal boundary layers are found to be a useful tracer in identifying hairpin vortex structures.

Large-scale motions in a turbulent boundary layer1. HYUNG JIN SUNG, JAE HWA LEE, KAIST — Direct numerical simulation of a turbulent boundary layer with $Re_{θ} = 2560$ was performed to investigate the spatially coherent structures associated with very large-scale motions (VLSMs). Inspection of the three-dimensional instantaneous fields showed that groups of hairpin vortices are coherently arranged in the streamwise direction and that these groups create significantly elongated low- and high-momentum regions with large amounts of Reynolds shear stress. Adjacent packet-type structures combine to form the VLSMs; this formation process is attributed to continuous stretching of the hairpins, coupled with lifting-up and backward curling of the vortices. We employed the modified feature extraction algorithm to identify the properties of the VLSMs of hairpin vortices. Patches with lengths greater than 3–44 account for more than 40% of all the patches and these VLSMs contribute approximately 45% of the total Reynolds shear stress. Finally, the application of linear stochastic estimation to the conditionally averaged flow field demonstrated the presence of packet organization in the form of a train of packets in the logarithmic layer.

Multi-scale geometry of flow structures in a flat-plate turbulent boundary layer . IVAN BERMEO-MORENO, Center for Turbulence Research, Stanford University, CALLUM ATKINSON, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University, SERGEI CHUMAKOV, Center for Turbulence Research, Stanford University, JULIO SORIA, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University, XIAOHUA WU, Royal Military College of Canada — We study the geometry of structures from the enstrophy and dissipation fields obtained from a DNS of a flat-plate turbulent boundary layer (J. Fluid Mech. 630, 5-41, 2009) following the non-local multi-scale methodology introduced in J. Fluid Mech. 603, 101-135, 2008. We compare the results with those of homogeneous isotropic turbulence. In the present analysis, geometric parameters are combined with physical quantities associated with the flow structures. Their evolution in time is studied through a series of snapshots obtained from the simulation, following a moving subdomain. Individual structures are tracked in time, relating their physical and geometric properties at the local and structure levels. The validity of two local identification criteria for the eduction of vortex tubess and sheets in wall-bounded flows is also evaluated.

Dynamic wall shear stress measurements in a turbulent channel flow . OMID AMILI, JULIO SORIA, Laboratory for Turbulence Research in Aerospace and Combustion, Department of Mechanical and Aerospace Engineering, Monash University, VIC (Australia), and Laboratory for Turbulence Research, Stanford University, CA (USA) — Shear stress is measured using a film-based shear stress sensor in turbulent channel flows. The sensor is used to measure the time-resolved local wall shear stress distribution in a turbulent channel flow. Measurements have been undertaken in a turbulent channel flow at Reynolds numbers up to 130,000 based on the bulk velocity and channel height. The measured fluctuating wall shear stress distribution provides spatio-temporal information of the characteristics of near wall structures by detecting their footprints. The span-wise extent of the positive two-point correlation of the stream-wise shear stress fluctuations provides the average width in the order of 100 wall units for the near-wall coherent structures. An investigation of the topological features of the velocity gradient and rate of strain tensors enables us to show an intrinsic characteristic of the near wall flow, which follows a two-dimensional flow pattern.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session EC Turbulence Modeling III Long Beach Convention Center 102A
4:10PM EC.00001 Study of the Invariants of the Velocity-Gradient Tensor in Homogeneous Isotropic Turbulence by means of 3C-3D Tomographic PIV, NICOLAS BUCHARMANN, SYLVAIN ROUVIER, JULIO SORIA, Monash University, Victoria 3800, Australia. NO TEAM — The study of coherent structures (CS) in turbulent flows is essential for understanding turbulence mechanisms in technological and theoretical relevant flows. The recent advent of instantaneous three-component and three-dimensional (3C-3D) measurement techniques now permits detailed experimental investigation into the dynamics and topology of CSs for by example analysis of the invariants of the velocity gradient tensor. For this purpose, the present work presents instantaneous, high-resolution 3C-3D Tomographic Particle Image Velocimetry (TPIV) measurements in a grid-generated, homogeneous isotropic turbulent flow (Re_λ ~140). The experiments are conducted in a larger water tunnel facility using a passive grid, four high-resolution digital cameras and a pulsed Nd:YAG laser for volume illumination. The invariants of the velocity gradient, rate of strain and rate of rotation tensor are used to characterize the dynamics and topology of the turbulent flow field and in particular its dissipation and vortex structure. Preliminary results are in agreement with previous literature and DNS simulations. The objective of this work is to measure these quantities experimentally and directly without additional assumptions pertaining to the structure and dynamics of the turbulent flow field.

4:23PM EC.00002 Investigation of Subgrid-Scale Turbulence in the Atmospheric Surface Layer using AHATS Field Data1, KHUONG NGUYEN, Clemson University, STEVEN ONCLEY, TOMAS HORST, PETER SULLIVAN, National Center for Atmospheric Research, CHENNING TONG, Clemson University — Data obtained in the atmospheric surface layer during the recent Advection Horizontal Array Turbulence Study (AHATS) field program are used to study issues met in large-eddy simulation (LES) of atmospheric boundary layer. The array technique, which has been successfully employed in several previous programs, is extended to include a second array to measure the advection of the subgrid-scale (SGS) stress. Pressure sensors are also deployed to measure the fluctuating pressure, enabling separation of the resolvable- and subgrid-scale pressure. We analyze the subgrid-scale terms in the joint probability density function (JPDF) of the resolvable-scale velocity, which must be reproduced by the SGS model in order for LES to predict correctly the resolvable-scale density function (JPDF). These terms include the conditional SGS stress (on the resolvable-scale velocity gradient tensor). The conditional SGS stress production rate, the conditional resolvable-scale pressure, and the conditional resolvable-scale pressure-strain rate correlation. We analyze the subgrid-scale terms in the joint probability density function (JPDF) of the resolvable-scale velocity, which must be reproduced by the SGS model in order for LES to predict correctly the resolvable-scale density function (JPDF). These terms include the conditional SGS stress (on the resolvable-scale velocity gradient tensor). The conditional SGS stress production rate, the conditional resolvable-scale pressure, and the conditional resolvable-scale pressure-strain rate correlation. We also analyze the advection and pressure terms in the SGS stress budget, which are important for understanding the dynamics of the SGS stress and for modeling the SGS stress using the transport equation.

3Supported by NSF

4:36PM EC.00003 Characterization of the flow on the axis of two confined-opposed-jets1, JEAN-FRANCOIS KRAWCZYNSKI, LUMINITA DANAILA, BRUNO RENOU, CORIA — We document an experimental investigation of a confined-opposed-jets flow, which is the basic spatial periodic pattern of a confined chamber. The question of the nature of the velocity fluctuations which are discussed is addressed. It is shown that the characteristic instabilities of this complex flow, along with the local confinement effects, generate large-scale quasi-organized fluctuations which are superimposed on the random/turbulent fluctuations. Small-scale statistics, such as the kinetic energy dissipation rate, are discussed and estimates based on inertial or large-scales methods are proposed and compared to traditional small-scales estimates. A scale-by-scale energy budget equation, analogous to the famous Yaglom’s equation for inhomogeneous and anisotropic turbulence is discussed and partially validated in this complex flow. It is shown that the energy transfer is mainly performed in planes perpendicular to the axisymmetry axis, whereas it is strongly inhibited over the axisymmetry direction.

1Financial support from French National Research Agency, Project Micromelange N-NT05-2 12482 is gratefully acknowledged.

4:49PM EC.00004 Scale-by-scale energy budget equations for large-eddy simulations, MICHAEL GAUDING, JENS HENRIK GOEBBERT, ROBERT FLESCH, CLAUDIA GUENTHER, NORBERT PETERS, RWTH-Aachen University — A detailed study of scale-by-scale energy budget equations of homogeneous shear turbulence is performed. The energy budgeted equations are formulated in terms of structure functions and involve the balance of turbulent kinetic energy production, energy transfer and energy dissipation. In the context of large-eddy simulation (LES) the transfer of energy towards small scales must be correctly satisfied in order to preserve statistically properties of the turbulent flow field. In this work a comparison of filtered DNS data with various LES models is performed in order to assess the performance of the models. Particular emphasis is laid on anisotropic effects. A closure for the subgrid term is proposed based on an eddy-viscosity ansatz. The analysis is performed by means of direct numerical simulations of homogeneous shear turbulence at Taylor-Reynolds number of 120. Homogeneous shear turbulence reveals effects such as large-scale vortex dynamics and anisotropy.

5:02PM EC.00005 Streamwise-constant model of intermittent turbulent pipe flow1, JEAN-LOUP BOURGUIGONN, BEVERLEY MCKEON, Caltech — A streamwise-constant model of intermittent turbulent pipe flow is presented, following the work on Couette flow of Gayme et al. 2010. The model consists of two evolution equations derived from Navier-Stokes, one for the streamfunction describing the in-plane velocities and one for the axial velocity. Under stochastic forcing, the model exhibits a quasi-periodic self-sustaining cycle characterized by the creation and subsequent decay of turbulent clusters of coherent structures remarkably similar to turbulent puffs. The flow structures inside these turbulent clusters correspond to the quasi-streamwise vortices and streaks observed in transition experiments. The time traces of the centerline velocity present numerous puff signatures, i.e. the centerline velocity drops suddenly and then increases smoothly nearly up to its laminar value. Under deterministic forcing, our model shows that the main features of turbulent pipe flow are robust and can easily be reproduced by solving a single momentum balance equation.

1The support of AFOSR grant # FA 9550-09-1-0701 (program manager John Schnitzer) is gratefully acknowledged.

5:15PM EC.00006 Eddy-Based Model for Wall-Bounded Turbulent Flows1, BRIAN ROSENBERG, ALEXANDER SMITS, Princeton University, SEAN BAILEY, University of Kentucky — Here we extend the wall-bounded turbulence model of Smits (2008). The original model identifies three eddy motions ubiquitous in wall-bounded flows and captures their energy scaling in wavenumber space. The coherent structures identified are near-wall longitudinal vortex-like streaks, the Large Scale Motions related to packets of hairpin vortices, and the Very Large Scale Motions, interpreted as either outer-layer bulges in boundary layers or meandering superstructures in internal flows. The three eddy functions are summed, neglecting nonlinear interactions, and the Reynolds stress behavior is obtained by integrating over all wavenumbers. While the original model utilizes Gaussian representations of the eddy motions in wavenumber space, we instead construct wavelet-based representations in physical space. The new eddy functions are expected to offer a better physical basis for modeling since the velocity signatures of simple eddies closely resemble wavelets. Simulations at various Reynolds numbers are then compared with the original model and experiments.

1Supported under AFOSR Grant FA9550-09-1-0569.
4:10PM ED.00001 Effect of Surface Roughness on heat transfer in a Turbulent Channel Flow, STEFANO LEONARDI, BENJAMIN CRUZ PEREZ, JOHN LUCENÁ, University of Puerto Rico at Mayaguez — DNSs are carried out for passive heat transport in a turbulent channel flow with surface roughness on the wall. The total heat transfer depends on the pitch to height ratio of the roughness. Several configurations have been studied, square bars, circular cylinders, V-shaped turbulators, segmented V-shaped turbulators, with and without fillets. A parametric study has been performed with the aim of finding the configuration with the largest heat transfer and the minimum drag. The effect of placing the roughness surfaces on both walls or only on one wall is also considered. For transverse square bars, ejections occurring on the leading edge of the roughness elements enhance the heat transfer. On the other hand, when V-shaped turbulators are placed on the wall a secondary motion is induced which is responsible of removing heat from the wall. Applications in turbine engines will be discussed at the conference.

4:23PM ED.00002 Turbulence modification by stable stratification in channel flow, MANUEL GARCIA-VILLALBA, University of Karlshue, JUAN C. DEL ALAMO, University of California San Diego. — We study the modification of the turbulent structure of plane channel flow under stable stratification through direct numerical simulations. The simulations are performed at moderate Reynolds numbers (up to Re = 550), in very large computational boxes (up to Lx = 60 h, Lz = 25 h where h is the channel half height) and considering a wide range of stratification levels (up to Ritu = 1920). For weak stratification or high Reynolds number, the turbulence is affected by buoyancy in the core of the channel but the near-wall region differs little from the neutral case. With increasing stratification, the near-wall streaks remain essentially unmodified while large-scale super-streaks are damped. In the central region internal gravity waves are dominant. In addition, there is an intermediate outer layer where the dynamics of the turbulent structures is governed by local fluxes. In this region energy spectra collapse when using local Obukhov scaling. With strong stratification, large laminar patches appear in the near wall region, and turbulent momentum and buoyancy fluxes vanish in the core of the channel. Linear transient-growth analysis of the mean velocity and density profiles reproduces the damping of the super-streaks in stratified channels, and predicts maximum energy amplification for infinitely-wide perturbations that are consistent with the internal waves observed in the simulations.

4:36PM ED.00003 DNS of multiscale-generated turbulence, JOHN CHRISTOS VASSILICOS, SYLVAIN LAIZET, Imperial College London — Four spatially evolving turbulent flows, one generated by a regular grid and three generated by fractal square grids of different aspect ratios, are studied through Direct Numerical Simulation (DNS). Method and an efficient domain decomposition is used in this study to perform such large simulations. Statistics such as turbulent intensities are investigated with the objective of analysing the two different regions (production and decay regions) downstream from the fractal square grids, as already observed in the experimental results of Mazellier & Vassilicos (PoF, 2010). The main goal of this numerical study is to identify the physical mechanisms implicated in the generation of turbulent flows, especially when generated at different scales, but also to compare the different levels of turbulence intensity generated by each grid.
4:49PM ED.00004 Use of POD to Investigate Large-Scale Structures in Turbulent Flow with Rib-Roughness
, MAZIAR E. SAMANI, DONALD BERGSTROM, Department of Mechanical Engineering - University of Saskatchewan — In this work, Large Eddy Simulation (LES) of turbulent Couette flow is used to study the flow structures associated with the hydrodynamic roughness created by a series of ribs of square cross-section mounted perpendicular to the flow. Due to their geometric simplicity, ribs have been previously used to simulate turbulent flow over a rough surface. Both direct numerical simulation (DNS) (Orlandi et al., 2006) and experimental methods (Krogstad et al., 2005) have been used to study this type of roughness in the context of pressure driven flow in a channel. In this study, LES is used to simulate turbulent Couette flow with rib roughness on one wall. Proper Orthogonal Decomposition (POD) is applied to determine the large-scale flow structures in the wake region of the roughness elements. The spanwise vorticity and both streamwise and wall-normal velocity components are used as the computational variables for the POD. Comparisons are made to a similar POD analysis of the wake structure of an infinite square cylinder mounted in a uniform flow. Of specific interest is the interaction of the rib wakes with the turbulent boundary layer above.

5:02PM ED.00005 Coherent Structures in the Flow over Two-Dimensional Dunes
, MOHAMMAD OMIDYEDEGANEH, UGO PIOMELLI, Queens University — Dunes are common large-scale bed irregularities found in rivers at high Reynolds numbers. One characteristic feature of the flow over dunes is the presence of boles, upwelling motions observed at the water surface. Understanding the dynamics of the eddies that cause boles is critical, since these eddies lift up sediment from the river bed and carry it to the surface. To analyse the dynamics of these large structures, we performed large eddy simulation on the flow over two-dimensional dunes at laboratory scale (the Reynolds number based on the average channel height and mean velocity is 18,900). Results show that the rollers generated in the separated shear layer interact with wall turbulence to form an inclined horseshoe vortex that reaches the surface; the upwelling is due to the ejection that occurs between the legs of the horseshoe. These vortices are initially vertical, but become inclined when they are advected to the surface; near-wall horseshoe vortices, on the other hand, start from a horizontal orientation. This indicates that these structures are unique to the flow over dunes, in which separation occurs at the crest. While the boil frequency is fairly low (they were found every 40k/Us time, approximately) they are known to affect sediment transport significantly. A more quantitative analysis of the contribution of large vortices to mass and momentum transport is presently being carried out.

5:15PM ED.00006 POD Study of Turbulent Boundary Layer Structures
, JON BALTZER, RONALD ADRIAN, Arizona State University, XIAOHUA WU, Royal Military College of Canada — 3D POD is performed on a recent DNS turbulent boundary layer simulation (Wu & Moin, Phys. Fluids, 2010). The modes of various scales are examined. Dominant modes include structures consistent with the inclined ramps commonly observed in turbulent boundary layers. POD modes are examined to interpret the spatial organizations of the structural elements.

5:28PM ED.00007 Development and Assessment of Proper Orthogonal Decomposition for Analysis of Turbulent Flow in Piston Engines
, K. LIU, D.C. HAWORTH, The Pennsylvania State University — High-resolution optical diagnostics (e.g., particle-image velocimetry – PIV) and high-resolution numerical models (e.g., large-eddy simulation – LES) are increasingly being used to develop advanced combustion systems for next-generation piston engines. To date, quantitative comparisons between PIV and LES for engines have been limited mainly to ensemble- (phase-) averaged mean quantities. Proper orthogonal decomposition (POD) has been proposed as an approach for analyzing the dynamics of complex in-cylinder processes, and as a basis for making objective quantitative comparisons between PIV and LES. Here three-dimensional, time-dependent datasets generated by performing multiple-cycle LES of motored flows for simple engine configurations are used to develop and assess the use of POD as a basis for the analysis of turbulent flow in piston engines. We explore POD variants that are required for analysis of statistically nonstationary flows in time-varying domains. We explore sensitivities of mode structure and convergence rate to spatial and temporal resolution, and perform comparisons of two-dimensional and three-dimensional POD analyses. And we explore the use of POD to identify and quantify cycle-to-cycle variations.

5:41PM ED.00008 Coherent turbulent motions in a Mach 3 boundary layer
, IZAAK BEEKMAN, YIN-CHIU KAN, STEPHAN PRIEBE, PINO MARTIN, University of Maryland — We examine coherent structures found in a Mach 3, compressible, turbulent boundary layer using a new, long domain (x = 500, y = 50), spatial direct numerical simulation (SDNS). Recent studies have shown that certain coherent motions, termed “superstructures,” or very large scale motions (VLSM), play an important dynamical role, strongly impacting the near wall cycle[1]. While most previous studies have been performed on incompressible boundary layer flows and at higher Reynolds numbers, Ringuette, Wu & Martin have shown that these structures are present at the conditions of the current simulation[2]. With this simulation we examine the dynamics and geometry of the large scale turbulence structures using statistical techniques, as well as visualizations. Additionally, we characterize the footprint of these coherent structures and their interaction with the wall.

5:54PM ED.00009 Methods for Large Eddy Simulation and Proper Orthogonal Decomposition of High Reynolds Number Turbulent Channel Flow
, K. LIU, D.C. HAWORTH, The Pennsylvania State University — High-resolution optical diagnostics (e.g., particle-image velocimetry – PIV) and high-resolution numerical models (e.g., large-eddy simulation – LES) are increasingly being used to develop advanced combustion systems for next-generation piston engines. To date, quantitative comparisons between PIV and LES for engines have been limited mainly to ensemble- (phase-) averaged mean quantities. Proper orthogonal decomposition (POD) has been proposed as an approach for analyzing the dynamics of complex in-cylinder processes, and as a basis for making objective quantitative comparisons between PIV and LES. Here three-dimensional, time-dependent datasets generated by performing multiple-cycle LES of motored flows for simple engine configurations are used to develop and assess the use of POD as a basis for the analysis of turbulent flow in piston engines. We explore POD variants that are required for analysis of statistically nonstationary flows in time-varying domains. We explore sensitivities of mode structure and convergence rate to spatial and temporal resolution, and perform comparisons of two-dimensional and three-dimensional POD analyses. And we explore the use of POD to identify and quantify cycle-to-cycle variations.

Sunday, November 21, 2010 4:10PM - 6:07PM – Session EE Instability and Turbulence: Couette and Channel Flow Long Beach Convention Center 102C

4:10PM EE.00001 Comparison of linear and nonlinear optimal perturbation transient growth in plane Couette flow
, S.M.E. RABIN, DAMTP, U. of Cambridge, C.P. CAULFIELD, BPI & DAMTP, U. of Cambridge, R.R. KERSWELL, Mathematics, U. of Bristol — Previous approaches to the question of transient growth have focused upon the study of linearised disturbances, with the assumption that it is the growth in the linear regime of linear optimal perturbations (LOPs) that nevertheless lead to a nonlinear regime and hence trigger the transition to turbulence. In this study we take a different approach by considering the full nonlinear problem. We look to extend the work considering pipe flow of Pringle (C. C. T. Pringle Ph.D. Bristol 2009) and use variational techniques to examine both the spatial structure and the normalised kinetic energy growth (gain) achieved by nonlinear optimal perturbations (NLOPs) in plane Couette flow. We show that in certain circumstances the gain achieved by the NLOP is significantly larger and has a noticeably different (and more complex) spatial structure from its counterpart LOP. We investigate the dependence on initial perturbation energy of the maximum predicted gain for selected Reynolds numbers and optimization times and propose that these inherently nonlinear structures may well be more significant in the transition to turbulence than LOPs.
4:23PM EE.00002 Streamwise vortices in shear flows: Stability of the lower branch states in Couette flow, PHIL HALL, SPENCER SHERWIN, Imperial College London — The relationship between asymptotic descriptions of vortex-wave interactions and more recent work on “exact coherent structures” is investigated. We have recently shown that the so-called “lower branch” state, which has been identified as playing a crucial role in these self-sustained processes, is a finite Reynolds number analogue of a Rayleigh vortex-wave interaction with scales appropriately modified from those for external flows to Couette flow. We show that the instability is concentrated in a layer which surrounds the critical layer and destroys the wave leaving the roll/streak flow to decay on a 1/R timescale.


4:36PM EE.00003 Streamwise vortices in shear flows: harbingers of transition and the skeleton of coherent structures, SPENCER SHERWIN, PHIL HALL, Imperial College London — A recent theoretical/numerical investigation (Hall & Sherwin 2010) demonstrates that vortex-wave interaction in the critical layer of a roll-streak system provides a driving mechanism that will maintain the coupled flow system in near-equilibrium. In addition, the theory predicts that rolls are maintained by tangential gradients of Reynolds stress within the critical layer. These results suggest some wall turbulence control strategies that are similar to chaos control methodology (Tardu, Chaos 2010).


4:49PM EE.00004 Critical layer structure in transitional Couette flows, HUGH BLACKBURN, Monash University, PHILIP HALL, SPENCER SHERWIN, Imperial College London — A recent theoretical/numerical investigation (Hall & Sherwin 2010) demonstrates that vortex-wave interaction in the critical layer of a roll-streak system provides a driving mechanism that will maintain the coupled flow system in near-equilibrium. In addition, the predictions made for the variation in the strength of the roll-wave system with Reynolds number asymptotically match those for the lower-branch states observed by Wang et al. (2007) in the Couette system. We use DNS of transitional Couette flows to examine two key predictions made by the theory. First, for fixed spanwise periodic wavelength, we examine the maximum streamwise periodic wavelength for which the flow does not relaminarize, which suggests a wave length below which equilibrium cannot be maintained. Second, we extract Reynolds stresses in the critical layer, and examine their relationship to observed roll structures, since the theory predicts that rolls are maintained by tangential gradients of Reynolds stress within the critical layer.


5:02PM EE.00005 Vorticity fluctuations in Plane Couette Flow, JOSE ORTIZ DE ZARATE, Universidad Complutense, JAN V. SENGERS, University of Maryland — In this presentation we evaluate the flow-induced amplification of the thermal noise in plane Couette configuration. The physical origin of the noise is the random nature of molecular collisions, that contribute with a stochastic component to the stress tensor (Landau’s fluctuating hydrodynamics). This intrinsic stochastic forcing is then amplified by the mode-coupling mechanisms associated to shear flow. In a linear approximation, noise amplification can be studied by solving stochastic Orr-Sommerfeld and Squire equations. We compare the efficiency of the different mechanisms, being the most important the direct coupling between Squire and Orr-Sommerfeld equations. The main effect is to amplify wall-normal vorticity fluctuations with a spanwise modulation at wave number around 1.5, a configuration that resembles the streaks that have been proposed as precursors of the flow instability.

5:15PM EE.00006 Nonlinear stability, bifurcation and resonance in granular plane Couette flow, PRIYANKA SHUKLA, PhD Student, MEHEBOOB ALAM, Prof. — A weakly nonlinear stability theory is developed to understand the effect of nonlinearities on various linear instability modes as well as to unveil the underlying bifurcation scenario in a two-dimensional granular plane Couette flow. The relevant order parameter equation, the Landau-Stuart equation, for the most unstable two-dimensional disturbance has been derived using the amplitude expansion method of our previous work on the shear-banding instability[1]. Two types of bifurcations, Hopf and pitchfork, that result from travelling and stationary linear instability modes, respectively, are analysed using the first Landau coefficient. It is shown that the nonlinear instability can appear in the linearly stable regime. The present bifurcation theory shows that the flow is subcritically unstable to disturbances of long wave-lengths ($k_x \rightarrow 0$) in the dilute limit, and both the supercritical and subcritical states are possible at moderate densities for the dominant stationary and traveling instabilities for which $k_x = O(1)$. We show that the granular plane Couette flow is prone to a plethora of resonances[2].


5:28PM EE.00007 Chaotic synchronization of the wall turbulence, SEDAT TARDU, LEGI, B.P. 53 X. 38041 Grenoble, France — Multiscale edge detection wavelet analysis is applied to the streamwise velocity fluctuations in the buffer layer through direct numerical simulations. The wavelet coefficients are rewritten using analytic signal approach to sort out their local amplitudes and wavenumbers. Large zones of approximately constant wavenumbers have been identified at different scales, and a parallelism is constructed between these observations and stochastic synchronization phenomena. The results we analyze strongly suggest that the wall turbulence is chaotically synchronized with the forcing induced by convecting coherent vortices near the wall, thus confirming our earlier results based on experimental velocity and wall shear stress time series (Tardu, Phys. Fluids, 2007). The spatial extent of phase-locked, synchronized zones feature a clear type I intermittency behaviour. The local amplitude intermittency in the synchronized zones is low, and the small-scale amplitude intermittency increases significantly when they are suppressed. The type I intermittency disappears in the viscous and log layers. These results suggest some wall turbulence control strategies that are similar to chaos control methodology (Tardu, Chaos 2010).
A nanohybrid structure is thus compared with SEM images of the cross-sections of the fibers. The nano-porous composite fibers produced by coagulation wet spinning were

Experimental data on the droplet shape/volume together with front propagation are compared with the solution of the model to extract information of the fiber's

In characterizing nano-porous fibers where the wicking front propagates through longitudinal micron size grooves and the liquid also penetrates inside the

The difference in groove to pore sizes produces a faster longitudinal than transversal

Contrary to what happens on a planar wetting substrate, a droplet

μ

in-plane water distribution of water in the PEFC components is quantitatively visualized by adopting image normalization method. The temporal evolution of

The objective of the present work is to characterize ionic transport through a carbon porous media in a capacitive cell correlating its transient electrical parameters with its ion absorption performance. For this, a constant and continuous power supply, and carbon aerogel were employed in three types of experiments. In the first type, the electrical assembly is submerged in a stationary solution and the parasitic elements of the electric circuit were estimated. In the second type, ion-saturated aerogel was used in a series of polarity inversion cycles to evaluate its behavior. Finally, in the third type, convective effects on the ion migration were evaluated when a constant flow rate is applied. For all the experiments, the total capacitor's charge was obtained from the experimental current data and compared to the amount of ions retained on the carbon media estimated by the variation in the aerogel's weight and in the solution's conductivity.

The objective of this study is to visualize the temporal evolution of water in a rotating channel on a horizontal line that moves

doing this, the synchrotron X-ray radiography with high spatial and temporal resolution is employed. X-ray images of

We present an experimental study of liquid-air flow inside a 500 x 500 x 1 mm square Hele-Shaw cell saturated with 0.1mm diameter glass spheres. The flow is characterized by using the light refraction at liquid-gas interface menisci as a marker to determine the time dependent position of the liquid-gas interface. In the flow analyzed, liquid motion is generated by partially filling the cell with water and then letting the water out through an outlet in the lower part of the cell. We have observed that in contrast to what occurs in a Hele-Shaw cell with no spheres where the interface is a horizontal line that moves downwards, for a Hele-Shaw cell filled with spheres, the liquid-gas interface is an irregular line that moves with localized sudden motions generated by surface tension effects occurring due the non-regular geometry of the sphere arrangements. The distance scale of these dynamic structures is approximately ten sphere diameters. These observations are potentially useful in the underground water flows and petroleum extraction.

Dynamics of poroelastic foams, YOEL FORTERRE, BENJAMIN SOBAC, IUSTI CNRS-Universite de Provence, Marseille, France — Soft poroelastic structures are widespread in biological tissues such as cartilaginous joints in bones, blood-filled placenta or plant organs. Here we investigate the dynamics of open elastic foams immersed in viscous fluids, as model soft poroelastic materials. The experiment consists in slowly compacting blocs of polyurethane solid foam embedded in silicon oil-tanks and studying their relaxation to equilibrium when the confining stress is suddenly released. Measurements of the local fluid pressure and foam velocity field are compared with a simple two-phase flow approach. For small initial compactions, the results show quantitative agreement with the classical diffusion theory of soil consolidation (Terzaghi, Biot). On the other hand, for large initial compactions, the dynamics exhibits long relaxation times and decompaction fronts, which are mainly controlled by the highly non-linear mechanical response of the foam. The analogy between this process and the evaporation of a polymer melt close to the glass transition will be briefly discussed.

The objective of this study is to visualize the temporal evolution of water in an operating (in situ) polymer electrolyte fuel cell (PEFC). To achieve this, the synchrotron X-ray radiography with high spatial and temporal resolution is employed. X-ray images of water inside individual PEFC components, such as the polymer membrane, gas diffusion layer (GDL), and endplate, are captured consecutively. As a result, the in-plane water distribution of water in the PEFC components is qualitatively visualized by adopting image normalization method. The temporal evolution of water in the anode GDL exhibits the back diffusion effect clearly. To examine the water accumulation phenomenon in the PEFC, X-ray μ-tomography method is adopted for visualizing the internal structure of GDL. The accumulation phenomenon seems to be attributed to the concentrated porosity in GDL structure. The water-saturation characteristics at the cathode GDL, including saturation time and speed, are found to be quite different from those at the anode GDL.

We present an experimental study of liquid-air flow inside a 500 x 500 x 1 mm square Hele-Shaw cell saturated with 1mm diameter glass spheres. We have observed that in contrast to what occurs in a Hele-Shaw cell with no spheres where the interface is a horizontal line that moves downwards, for a Hele-Shaw cell filled with spheres, the liquid-gas interface is an irregular line that moves with localized sudden motions generated by surface tension effects occurring due the non-regular geometry of the sphere arrangements. The distance scale of these dynamic structures is approximately ten sphere diameters. These observations are potentially useful in the underground water flows and petroleum extraction.

In the last few years, the transport of ionic species through a porous media and an electric field has been subject of intense research because of its many applications. The objective of the present work is to characterize ionic transport through a carbon porous media in a capacitive cell correlating its transient electrical parameters with its ion absorption performance. For this, a constant and continuous power supply, and carbon aerogel were employed in three types of experiments. In the first type, the electrical assembly is submerged in a stationary solution and the parasitic elements of the electric circuit were estimated. In the second type, ion-saturated aerogel was used in a series of polarity inversion cycles to evaluate its behavior. Finally, in the third type, convective effects on the ion migration were evaluated when a constant flow rate is applied. For all the experiments, the total capacitor's charge was obtained from the experimental current data and compared to the amount of ions retained on the carbon media estimated by the variation in the aerogel's weight and in the solution's conductivity.
5:15PM EF.00006 DNS of inertial flows in porous media: Assessment of mesh quality and resolution1, JUSTIN FINN, SOURABH APTE, BRIAN WOOD, Oregon State University — At modest flow rates ($10 < Re \leq 300$) through porous media and packed beds, fluid inertia can result in complex steady and unsteady recirculation regions, dependent on the local pore geometry. We present methods to parameterize and simplify mesh generation for packed beds, with an eye toward obtaining efficient mesh independence for Reynolds numbers in the inertial and unsteady regimes. To handle the geometric singularity at the sphere-sphere and sphere-wall contact points, we use a fillet bridge model, in which every pair of contacting entities are bridged by a fillet, eliminating a small fluid region near the contact point. A second order accurate, parallel, incompressible flow solver [Moin and Apte, AIAA J. 2006] is used to simulate flow through three different sphere packings: a periodic simple cubic packing, a wall bounded hexagonal close packing, and a randomly packed tube. Mesh independence is assessed using several measures including Ergun pressure drop coefficients, viscous and pressure components of drag force, kinetic energy, kinetic energy dissipation and interstitial velocity profiles. Progress toward large scale simulations of flow through randomly packed $10^3$ pores will be discussed.

1This work is funded by NSF-CBET award number 0933857. JF and SA also acknowledge DoE-NETL’s URS contract and ORISE program.

5:28PM EF.00007 Pressure-driven flow in a channel with porous walls1, QIANLONG LIU, Johns Hopkins University, ANDREA PROSPERETTI, Johns Hopkins University and University of Twente, The Netherlands — The finite-Reynolds-number three-dimensional flow in a channel bounded by one and two parallel porous walls is studied numerically. The porous medium is modelled by spheres in a simple cubic arrangement. The results for the slip velocity at the surface of the porous layers are compared with the phenomenological Beavers-Joseph model. It is found that the value of the slip coefficient is different for pressure-driven and shear-driven flow. A modification of the relation is suggested to deal with this feature. Furthermore, detailed results on the flow structure and the hydrodynamic forces and couple acting on the sphere layer bounding the porous medium are reported and their dependence on the Reynolds number illustrated. It is shown that, at finite Reynolds numbers, a lift force acts on the spheres, which may be expected to contribute to the the mobilization of bottom sediments.

1This study was supported by National Science Foundation under grants CBET-0625138 and CBET-0754444.

5:41PM EF.00008 Modeling Deformable Fibrous Media Using Direct Numerical Simulations. IRFAN KHAN, CYRUS AIDUN, Georgia Institute of Technology — A micro-mechanical approach is used in this work to investigate the behavior of deformation in saturated fibrous media. The geometry of the porous media is approximated using model geometry made of cylinders in orthogonal arrangement with appropriate boundary conditions. The approach is based on direct numerical simulation that uses a hybrid lattice Boltzmann and finite element method for modeling the fluid and solid phases respectively. It has already been shown that the macroscopic behavior of real porous media can be recovered using model geometry as long as the parameters, porosity, permeability and compressive modulus are matched. Thus based on these critical parameters it is found that cylinders in skewed orthogonal arrangement behave as real layered fibrous porous media during saturated compression. Further an analytical expression is developed to predict the compressive modulus of orthogonal arrangement of cylinders. The expression shows that there is no direct effect of fiber diameter on the compressive modulus of such arrangements, which is also confirmed through direct numerical simulations.

5:54PM EF.00009 Fingering Instabilities during Capillary Imbibition into Paper, N. YOUNG, B.G. HIGGINS, W.D. RISTENPART, Dept. Chem. Engr. & Mat. Sci., Univ. California at Davis — When a porous medium, such as a piece of paper, is placed into contact with a liquid reservoir, capillary action drives the liquid through the porous medium. The penetration distance $L(t)$ of the liquid/air interface is typically described by the Lucas-Washburn equation, with any deviations normally occurring on a length scale set by the average pore size. Here we report that solutions with a sufficient amount of hydrophobic solute undergo a fingering instability during capillary imbibition into paper. The finger amplitudes are two to three orders of magnitude larger than the average pore size, suggesting that the typical capillary fingering mechanism is not operative. Instead, we demonstrate that the finger growth rate is directly proportional to the solute concentration, and is strongly mediated by the ambient relative humidity. We interpret the fingering in terms of an instability driven by solutal effects on the local imbibition velocity, and we discuss the implications for various applications including thin layer chromatography and paper-based microfluidics.

6:07PM EF.00010 Some Remarks on the flow of Viscoelastic Fluids in Porous Media, PURNA KALONI, Dept.of Math. and Stat. — Flow of porous media plays important roles in many branches of science and engineering. Because of the complications involved, studies in porous media have, largely, been experimental and the progress in theoretical modeling has been very slow Thus the one dimensional empirical model of Darcy, proposed in 1856, was extended to a non-linear empirical model by Forcheimer in 1901, and a diffusive term was added by Brinkman in 1949. In sixties and seventies, Whitaker, Slattery and Lundgren applied volume averaging technique to Navier-Stokes equation and gave heuristic account of the above models. Apart from some minor issues, the flow of viscous fluids in porous media is now well understood. This is, however, not the case in viscoelastic fluid flows in porous media. The empirical models are being employed without recognizing their empirical nature. Linear models are being used which do not reduce to the viscous model as the elastic parameters are set equal to zero. There are serious issues with the averaging process. Our purpose is to elaborate on the above problems and hopefully, suggest a reasonable model equation.

Support was given by the National Science Foundation Project OCE-0726339.

Sunday, November 21, 2010 4:10PM - 6:20PM — Session EG GFD: Oceanography II — Long Beach Convention Center 103B

4:10PM EG.00001 Downwelling in Boundary Currents Subject to Buoyancy Loss1, CLAUDIA CENEDESE, Woods Hole Oceanographic Institution — Recent observational, theoretical, and modeling studies all suggest that the upper part of the downwelling limb of the thermohaline circulation is concentrated in strong currents subject to buoyancy loss near lateral boundaries. This is fundamentally different from the traditional view that downwelling takes place in regions of deep convection. Theoretical understanding of what controls the downwelling near boundaries (its magnitude, length scales) is weak and relies on parameterizations of poorly known turbulent mixing processes. In this study, laboratory experiments were carried out to explicitly resolve the turbulent mixing due to convective plumes and identify where downwelling takes place. The dependence of the downwelling, circulation, and free surface temperature on the non-dimensional parameters that describe the boundary current and surface forcing will be discussed. The laboratory results are compared to previous theories and numerical models for downwelling boundary layers, to determine whether the details of these small-scale turbulent processes need to be explicitly resolved in order to represent their influence on the larger-scale circulation.

1Support was given by the National Science Foundation Project OCE-0726339.
4:23PM EG.00002 Diffusion-limited settling of porous particles in a stratified fluid. KOLJA KINDLER, BO LIU, Max Planck Institute for Marine Microbiology. ROMAN STOCKER, Massachusetts Institute of Technology, ARZHAND KHALILI, Max Planck Institute for Marine Microbiology — Marine particles settle at low Reynolds numbers, are often highly porous, and are frequently observed to accumulate at pycnoclines. We present the first study of the settling of porous particles in a stratified fluid, by combining laboratory experiments and Lattice-Boltzmann simulations. We find that porosity markedly affects settling by causing retention of particles at pycnoclines. The excess density of highly porous particles is largely determined by the density of the interstitial fluid. The latter is adaptive in a stratified ambient, through exchange with the surrounding fluid. For low-permeability particles at sharp density interfaces, we observed the retention time to scale quadratically with particle size, as predicted based on a purely diffusive exchange of interstitial fluid. The simulations reveal that the interstitial fluid exchange, and thus the settling velocity, is modulated by a wake of lighter fluid that the particle entrains from upper layers. Both porosity and wake entrainment contribute to increase drag. These findings will affect estimates of carbon export rates from the upper ocean to the deep sea.

4:36PM EG.00003 LES of turbulent stratified flows on shallow continental shelves. GUILLAUME MARTINAT, Old Dominion University, Norfolk, VA, ANDRES TEJADA-MARTINEZ. University of South Florida, Tampa, FL, CHESTER GROSCH, Old Dominion University — Shelf slope length. The baroclinic energy flux, turbulence production and turbulent dissipation rate increase with increasing velocity depending on the height above the bottom. The maximum turbulent kinetic energy, dissipation and production lag with respect to the peak external near-critical region of the topography. Nonlinear processes become important in the vicinity of the slope. The propagating internal tide has higher harmonics, and afterward it effects on the intensification rate. The maximum turbulent kinetic energy, dissipation and production lag with respect to the peak external near-critical region of the topography. Nonlinear processes become important in the vicinity of the slope. The propagating internal tide has higher harmonics.

4:49PM EG.00004 Turbulent Mixing Efficiency in Stratified Couette Taylor Flow. BRUCE RODENBORN, GUENTHER EBERT, HARRY L. SWINNEY. Center for Nonlinear Dynamics, Dept. of Physics, UT Austin — Ocean mixing is critical to sustaining the meridional overturning circulation. Global ocean and climate models must parameterize ocean mixing because it occurs at scales well below the resolution of the models. The current understanding of ocean mixing requires about 20% of the kinetic energy in a turbulent flow to be converted into a change in the fluid’s gravitational potential energy. Laboratory work on mixing efficiency has used towed grids and other means to create turbulence, but this turbulence is not sustained and its relation to the turbulent patches observed in the oceans is not known. We study mixing in a linearly stratified fluid contained between two counter-rotating cylinders. An initial density variation of up to 200% over the height of our system is achieved using sodium polytungstate salt. Measurements are made for laminar to fully turbulent flows - Reynolds numbers from a few hundred to 10,000. The flow pattern is visualized using Kalliroscope, and the characteristic vertical length scale is determined from spatial Fourier transforms of images. The power input is determined by measuring the torque and rotation rate of both cylinders. The fluid’s gravitational potential energy is determined by measuring density as a function of height. We find that mixing efficiency is strongly dependent on the total Reynolds number and the total initial density variation.

5:02PM EG.00005 Multiple-Scale Asymptotics for Oceanic Fluid Dynamics: Coupled Planetary- and Quasi-Geostrophic Equations1. IAN GROOMS, KEITH JULIEN, University of Colorado, Department of Applied Mathematics, BAYLOR FOX-KEMPER, University of Colorado, CIRES — The planetary geostrophic (PG) equations for large-scale oceanic flow are linked to the quasigeostrophic (QG) equations for mesoscale flow in a multiple-scales asymptotic expansion. The models describe the coupling of planetary-scale and mesoscale dynamics: eddy kinetic energy is generated by baroclinic instability of the planetary flow, and the resulting eddy buoyancy fluxes feed back on the planetary flow. Anisotropy of the planetary flow is seen to play a key role in allowing the two-way coupling. The resulting equations are amenable to theoretical and computational investigation of the interaction of mesoscale and planetary scale dynamics.

5:15PM EG.00006 Evaluation of turbulent Prandtl (Schmidt) number parameterizations for stably stratified turbulent flows1. ZACHARY ELLIOTT2, SUBHAS VENAYAGAMOORTHY. Colorado State University — In this study, we evaluate four different formulations of the turbulent Prandtl (Schmidt) number Prt = νt/Γt, where νt is the eddy viscosity and Γt is the scalar eddy diffusivity, for stably stratified flows. All four formulations of Prt are strictly functions of the gradient Richardson number Ri which is a measure of the strength of the stratification. A zero equation turbulence model for the eddy viscosity νt in a one-dimensional, turbulent channel flow is considered to evaluate the behavior of the different formulations of Prt. Both uni-directional and oscillatory flows are considered to simulate conditions representative of practical flow problems such as atmospheric flows and tidally-driven estuarine flows, to quantify the behavior of each of the four formulations of Prt. We discuss which of the models of Prt allow for a higher rate of turbulent mixing and which models significantly inhibit turbulent mixing in the presence of density stratification. The basis underlying the formulation of each model in conjunction with the simulation results are used to highlight the importance of choosing the appropriate parameterization of Prt, given a model for νt in for stably stratified flows.

5:28PM EG.00007 Response of a stable stratified jet to surface wind and buoyancy forcing. HIEU PHAM, SUTANU SARKAR. University of California, San Diego — The fine-scale response of a subsurface linearly-stratified jet to the forcing of surface wind stress and surface cooling (downward buoyancy flux) is investigated using Direct Numerical Simulation. The simulation involves a symmetric jet situated below a laminar surface layer driven by a constant windstress. The surface layer is well mixed while the jet is stably stratified such that the gradient Richardson number is larger than the critical value for linear shear instability. The simulation setup follows the background conditions in the Equatorial Undercurrents (EUC) and, similar to the EUC, internal waves and intermittent patches of intense dissipation are observed in spite of nominally stable conditions. However, the wave momentum flux is significantly smaller than the Reynolds turbulent stress extracted from the background surface forcing. The wave energy flux is also smaller than the turbulent production. Intermittent patches of intense turbulence are observed inside the jet upper-flank where the background gradient Richardson number is larger than 0.25. The dissipation rate inside the patches is at least three orders of magnitude larger than the ambient value. The patches are the results of ejections of fluid parcels into gravitationally unstable regions. The ejections are observed to direct both upward and downward and are driven by the formation of vortex tubes.

5:41PM EG.00008 Bottom turbulence during resonant generation of Internal waves at a critical slope. BISHAKHDATT GA YEN, SUTANU SARKAR, UCSD — A numerical study based on direct numerical and large eddy simulation is performed to investigate internal tide generation that occurs when a barotropic tide oscillates over a sloping bottom. An intense boundary flow is generated in the near-critical case with slope angle equal to the natural internal wave propagation angle. The intensification of upslope and downslope flow increase with the length of the near-critical region of the topography. Nonlinear processes become important in the vicinity of the slope. The propagating internal tide has higher harmonics, subharmonics and inter harmonics. The resonant wave undergoes both convective and shear instability and promote strong turbulence over the entire slope and afterward it effects on the intensification rate. The maximum turbulent kinetic energy, dissipation and production lag with respect to the peak external velocity depending on the height above the bottom. The baroclinic energy flux, turbulence production and turbulent dissipation rate increase with increasing slope length.
5:54PM EG.00009 Drift in sheared shallow water waves\textsuperscript{1}, WILLIAM R.C. PHILLIPS, Swinburne University of Technology, ALBERT DAI, KUAN TJIAN, University of Illinois at Urbana Champaign — The drift in an \( \alpha \) monochromatic wave field on a shear flow whose characteristic velocity is \( \theta \) smaller than the phase velocity of the waves is considered. It is found that shear plays an increasingly important role as the depth decreases. Details of the shear flow likewise affect the drift. Two temporal cases common in coastal waters are studied: wind driven shear and current driven shear. In the former, the magnitude of the drift (maximum minus minimum) in shallow water waves is increased significantly above the Stokes drift. In the latter, on the other hand, the magnitude decreases. However, while the drift at the free surface is oriented always in the direction of wave propagation in stress driven shear, that is not always the case in current driven shear, especially in long waves as the boundary layer grows to fill the layer. This later finding is of particular interest vis a vis Langmuir circulation, which arise through an instability that requires differential drift and shear of the same sign. This means that while Langmuir circulation form near the surface and grow downwards (top down), perhaps to fill the layer, in stress driven shear, their counterparts in current driven flows grow from the sea floor upwards (bottom up) but can never fill the layer.

\textsuperscript{1} NSF-OCE-0116921

6:07PM EG.00010 Coherent structures in a stratified and rotating shear layer with horizontal shear\textsuperscript{1}, ERIC AROBONE, SUTANU SARKAR, University of California, San Diego — One of the least understood scales of the ocean is the submesoscale. Here, rotation is important but does not necessarily control the dynamics, instabilities and nonlinear cascades are possible, and stable stratification affects the flow. Previous work by the authors has revealed the zigzag instability, coherent structures, and strong vertical mixing for turbulent horizontal shear flow in a stratified medium. Prior investigations into the rotating shear layer have shown the formation of longitudinal vortices and destabilization for weak anticyclonic rotation and two-dimensionlization for cyclonic and strong anticyclonic rotation. We will perform numerical experiments to examine the effect of both rotation and stratification on coherent dynamics with environmental parameters appropriate for submesoscale flows. Competition between the increased vertical correlations associated with rotation and decreased vertical correlations associated with stratification will be assessed. Additionally the statistical evolution and physical mechanisms driving the flow evolution will be discussed.

\textsuperscript{1}Supported in part by NSF 0835839

Sunday, November 21, 2010 4:10PM - 6:20PM – Session EH Convection and Buoyancy Driven Flows III Long Beach Convention Center 103C

4:10PM EH.00001 Direct Actuation of Small-Scale Motions for Enhanced Heat Transfer in a Straight Channel\textsuperscript{1}, PABLO HIDALGO, ARI GLEZER, Georgia Institute of Technology — Heat transfer enhancement by small-scale flow interactions that are induced within the core flow of a heated, high-aspect ratio straight channel are investigated experimentally. Direct actuation of small scale motions is provided by streamwise-embedded piezoelectrically-driven cantilevered reeds that span the entire channel height. Deliberate interactions between the reeds and a given core flow lead to the formation of time-periodic vorticity concentrations over a range of vibration frequencies that are advected with the core flow and induce small-scale motions near the channel’s surfaces. Heat transfer measurements are obtained using novel, microfabricated heaters with integrated temperature sensors that are deposited on a silicon substrate. It is shown that the actuation disrupts the thermal boundary layers and result in a significant enhancement of the local and global heat transfer along the channel compared to the baseline (unactuated) flow. The interactions between the reed-induced motions and the channel internal surfaces and mixing within the core flow are investigated in detail using high resolution particle image velocimetry (PIV) with emphasis on local and global heat transfer across the channel boundaries.

\textsuperscript{1}Supported by DARPA and UTRC.

4:23PM EH.00002 Influence of temperature-dependent fluid properties on heat transfer in turbulent channel flow \textsuperscript{1}, ALFREDO SOLDATI, FRANCESCO ZONTA, CRISTIAN MARCHIOLI, Dept. Energy Technologies, University of Udine — Forced-convection heat transfer in turbulent liquid flows is parametrized by a correlation between the Nusselt number, \( Nu \), the flow Reynolds number, \( Re \), and the Prandtl number, \( Pr \). Most of existing expressions for such \( Nu = f(Re, Pr) \) correlation were developed under the assumption of negligible dependence of the thermo-physical properties of the liquid on temperature. This may be a bottleneck when system optimization is required. In this work we use pseudo-spectral direct numerical simulation to investigate the influence of temperature-dependent fluid properties on the overall turbulent heat transfer. In particular, we focus on turbulent channel flow of water, and we let viscosity vary with temperature at fixed \( Pr \). Compared to the case of constant thermo-physical properties, it is observed that, already at low \( Re \), temperature-dependent variations alter velocity profiles, and modify both the Nusselt number and the friction factor significantly (up to about 10 %).

4:36PM EH.00003 Viscous Lock-Exchange in Rectangular Cells: Calculation and Experiments \textsuperscript{1}, DOMINIQUE SALIN, University Pierre and Marie Curie, JEROME MARTIN, CNRS, NICOLE RAKOTOMALALA, University Pierre and Marie Curie, LAURENT TALON, CNRS — In a viscous lock-exchange gravity current, which describes the reciprocal exchange of two fluids of different densities in a horizontal channel, the front between two Newtonian fluids spreads as the square root of time. The resulting diffusion coefficient reflects the competition between the buoyancy driving effect and the viscous damping, and depends on the geometry of the channel. This lock-exchange diffusion coefficient has already been computed for a porous medium, a 2D Stokes flow between two parallel horizontal boundaries separated by a vertical height, \( H \), and, recently, for a cylindrical tube. In this presentation, we calculate it, analytically, for a rectangular channel (horizontal thickness \( b \), vertical height \( H \)) of any aspect ratio \( (H/b) \) and compare our results with experiments in horizontal rectangular channels for a wide range of aspect ratios \( (1/10 - 10) \). We also discuss the 2D Stokes-Darcy model for flows in Hele-Shaw cells and show that it leads to a rather good approximation, when an appropriate Brinkman correction is used. An extension to the case where the density contrast between the two fluids is generated by a chemical reaction is also discussed.

4:49PM EH.00004 Natural Convection in a Slot Subject to a Spatially Distributed Heating \textsuperscript{1}, M.Z. HOSSAIN, JERZY M. FLORYAN, University of Western Ontario — Natural convection in a fluid contained in an infinite horizontal slot subject to a spatially distributed heating has been investigated for a wide range of Prandtl numbers \( Pr \). Detailed results are presented for the case of the lower wall subject to heating being a sinusoidal function of one of the coordinate, with its spatial distribution described by the heating wave number \( \alpha \) and its intensity expressed in terms of the Rayleigh number \( Ra \). The primary response of the system, which represents a forced response, consists of convection in the form of rolls whose structure is determined by the particular values of \( Ra \) and \( \alpha \). Linear stability of convective motion has been considered and conditions leading to the emergence of roll instability have been identified. Two mechanisms of instability motion at the onset have been identified. In the case of moderate \( \alpha \) the pattern of instability is generally locked-in with the pattern of heating according to the relation \( \delta_{\alpha} = \alpha/2 \). In the case of large \( \alpha \), the critical disturbance wave number approaches value \( \delta_{\alpha} = 1.56 \) and the fluid response is similar to that found in the case of a uniformly heated wall. The first mechanism dominates if the spatial modulation of the flow is sufficiently strong while the second one dominates in the case of weak spatial modulation.
Three-dimensional extraction and analysis of thermal plumes in turbulent Rayleigh-Bénard convection

MATTHIAS KACZOROWSKI, The Chinese University of Hong Kong, OLGA SHISHKINA, German Aerospace Center, KE-QING XIA, The Chinese University of Hong Kong — We report a new method for extracting the thermal plumes (TPs) in turbulent Rayleigh-Bénard convection (RBC) which allows us to analyze the properties of the TPs and the background fluid. The investigation is based on direct numerical simulations of RBC in a cube \((8 \times 10^4 \leq Ra \leq 5 \times 10^7)\) filled with fluid of Prandtl number \(Pr = 4.38\). The basis of our extraction method is the idea that plumes convectively transport heat through the fluid, so that a heat flux threshold can be employed to extract the TPs. It is shown that this method yields reliable results over a wide range of \(Ra\) and at any vertical wall distance \(z\). Characteristic quantities of the flow are investigated on the boundaries of the TPs and the mean properties of the TPs and the background fluid are investigated as a function of the vertical coordinate. The \(Ra\)-scaling of a characteristic length scale of the TPs is examined and compared to that of the thermal boundary layer thickness.

Work supported by the Research Grants Council of Hong Kong SAR (Project No. CUHK 403807) and the EU Science and Technology Fellowship China.

Prandtl-Blasius temperature and velocity boundary layer profiles in turbulent Rayleigh-Bénard convection

KE-QING XIA, The Chinese University of Hong Kong, QUAN ZHOU, Shanghai University, RICHARD STEVENS, University of Twente, KAZUYASU SUGIYAMA, University of Tokyo, SIEGFRIED GROSSMANN, Philips-Universitäts-Marburg, DETLEF LOHSE, University of Twente — The shapes of the velocity and temperature profiles near the horizontal conducting plates’ center regions in turbulent Rayleigh-Bénard convection are studied numerically and experimentally over the Rayleigh number range spanning from \(10^3\) to \(3 \times 10^{11}\) and the Prandtl number range 0.7–5.4. The results show that both the temperature and velocity profiles well agree with the classical Prandtl-Blasius laminar boundary-layer profiles, if they are re-sampled in the respective dynamical reference frames that fluctuate with the instantaneous thermal and velocity boundary-layer thicknesses. The study further shows that the Prandtl-Blasius boundary layer in turbulent thermal convection not only holds in a time-averaged sense, but is also valid in an instantaneous sense most of the time.

Numerical Simulations of Rapidly Rotating Convection with Boundary Topography

MICHAEL CALKINS, JEROME NOIR, JEFF ELDREDGE, JON AURNOU, University of California, Los Angeles — The Earth’s magnetic field is generated by vigorous convective motions in the molten iron outer core. Observations over the last 400 years show that while many of the morphological features of the geomagnetic field change over time, others appear to have remained relatively constant. In addition, investigations of the Earth’s rotation rate, or length of day (LOD), show that the mantle and core are strongly coupled. One possible mechanism to explain the geomagnetic field and LOD observations is the interaction of convective motions with topographic features at the core-mantle boundary (CMB). To examine the effects of CMB topography on the dynamics of the Earth’s core, we present results from a suite of quasi-geostrophic, thermal convection simulations with boundary topography. The primary effects of the topography are an increase in heat flow and zonal flow magnitude. We find that the topography leads to the formation of closed streamlines in the lee of the topography, bearing resemblance to the structures observed in the geomagnetic field. Furthermore, the effects of the topography become more pronounced as the Ekman number is reduced, suggesting CMB topography may be important in controlling the convective dynamics in the core.

Plumes and Fountains in a Cross-Wind

BRUCE SUTHERLAND, ALEXENDRA ANDERSON-FREY, JOSEPH ANSONG, University of Alberta — Sour gas flares attempt to burn off hydrogen sulfide (H2S), a poisonous gas that can kill at concentrations higher than 1000PPM. Because in some areas of the Rocky Mountain foothills the concentration of the gas before burning can be as high as 20%, flaring must be extremely efficient to prevent disaster. Recent studies have shown that cross-winds can reduce the efficiency to 30% meaning the concentration of unburned gas at the source can be as high as 60000PPM. Engineers rely on atmospheric dispersion to reduce the concentration to tolerable levels before the plume extends to the ground. To predict the dispersion of the gas close to the source, the US Environmental Protection Agency uses a numerical model, AERMOD, that heuristically adapts plume theory to account for the effects of winds and atmospheric inversion. They do not account for the fact that H2S is heavier than air at room temperature and so would tend to pool in valleys after cooling. We have performed laboratory experiments to examine the dynamics of positively and negatively buoyant plumes in uniform and stratified environments with a uniform background flow. The results are then compared with the predictions of the AERMOD model.

Experimental study of natural convection inside a differentially heated enclosure with internal heat generation

COLIN BUTLER, MARCO GIRON, DAVID NEWPORT, University of Limerick — This study is motivated by the use of natural convection correlations in the early stages of thermal design. While correlations are widely available for benchmark geometries, in practice compartments may have many heated surfaces and several heat generating objects. An experimental investigation is undertaken to examine the influence of cavity differential heating on the natural convection flow from an isothermal circular horizontal cylinder. The square compartment, of length \(L\), contains the centrally positioned cylinder of diameter \(0.1L\). The vertical walls are differentially heated, while the remainders are assumed adiabatic. Steady-state temperature measurements were taken at 15 different locations inside the cavity. The air flow fields and velocities were measured using a 2D PIV system. Results are presented in the form of Nusselt number correlations, velocity vector maps and boundary layer profiles for different values of the Rayleigh number and temperature difference ratio \((T^*)\). A circular airflow was observed inside the compartment. The plume rising from the cylinder interferes with this stream with varying results depending on \(Ra\) and \(T^*\). The flow structures become increasingly dominated by the presence of the cylinder with increasing \(Ra\) and \(T^*\) despite the Grashof number for the cylinder being several orders of magnitude lower than that for the cavity.

Buoyancy-Induced Columnar Vortices for Power Generation

MARK W. SIMPSON, ARI GLEZER, Georgia Institute of Technology — Large-scale inherent instability of a thermally stratified, solar-heated air layer is exploited for power generation by deliberately enhancing the formation of intense columnar vortices such that each vortex drives a vertical-axis turbine. In nature, buoyancy-driven vortices (“dust devils”) occur spontaneously, with core diameters of 1-50 m at the surface, heights up to one kilometer, with induced air flow of considerable angular and linear momentum. Meter-scale laboratory experiments have demonstrated the nucleation and sustainment of strong buoyancy-driven vortices over a plane heated surface driven by a controllable power source. The present investigation focuses on the characterization of the columnar vortex and passive control of its core structure and strength for harvesting mechanical energy. It is shown that vortices having cores with nearly-uniform vorticity distributions can be “anchored” to small ground protrusions, and their circulation and angular momentum can be controlled by geometrical modifications of these surface protrusion and simple flow vanes.
4:10PM EJ.00001 A proportional-integral-differential control of flow over a circular cylinder\textsuperscript{1}. DONGGUN SON, SEUNG JEON, HAEBEOL SEOM, Seoul National University — In the present study, we apply a proportional(P)-integral(I)-differential(D) feedback control to flow over a circular cylinder for suppression of vortex shedding in the wake. The transverse velocity at a centerline location in the wake is measured and used for the feedback control. The sensing location is varied from 1d to 4d from the center of the cylinder. The actuation (blowing/suction) is provided to the flow at the upper and lower slots on the cylinder surface near the separation point based on the P, PI or PD control. Given each sensing location, the optimal proportional gain in the sense of minimizing the sensing velocity fluctuations is obtained for the P, PI and PD control. The additions of I and D controls to the P control certainly increase the control performance and broaden the effective sensing location. The P, PI and PD controls significantly reduce the velocity fluctuations at sensing locations and attenuate vortex shedding in the wake, resulting in the reductions in the mean drag and lift fluctuations.

\textsuperscript{1}Supported by the NRL and WCU Programs of KRF, MEST, Korea.

4:23PM EJ.00002 Active control of vortex induced vibrations for flow past a circular cylinder, M. SRIDHAR, B.S.V. PATNAIK, IIT Madras, Chennai - 600 036 — The flow past rigid bodies which are flexibly mounted would experience vortex induced oscillations. The phenomenon of eddy shedding is responsible for the asymmetric force distribution on the cylinder. In a variety of natural and technological applications, these vortex induced oscillations may cause resonance and structural failure [1]. Controlling vortex shedding by active annihilation of the wake vortices is of interest in flow control studies [2]. An active closed loop feedback control algorithm is designed and implemented for controlling the flow induced oscillations in a flexible cylinder. The control is achieved by blowing/suction from two sides of the cylinder, and it is integrated along with the equations for mass, momentum transport. The state of the flow is reported through multiple sensors and the quantum of actuations is performed by the controllers as dictated by the control algorithm. Present simulations are carried out at low Reynolds number, 100 and 200, and the complete suppression of self-excited oscillations is observed.


4:36PM EJ.00003 The Hydrodynamic Cart-Pole: Experiments in Machine Learning and Control of Fluid-Body Interactions, JOHN W. ROBERTS, JACOB STEINHARDT, MIT - CSAIL, SAVERIO SPAGNOLIE, UCSB, RUSS TEDRAKE, MIT - CSAIL — Unsteady fluid-body interactions at intermediate Reynolds numbers exhibit a great deal of dynamical complexity, as well as a great deal of structure. Abundant evidence from nature demonstrates that this structure can be exploited to achieve high performance at dynamical tasks. In this talk, we present experimental work on a simple fluid-body system, a hydrodynamic analogue to the well-studied “cart-pole” system. Examples include balancing an immersed robot at a passively unstable equilibrium, as well as more fundamentally nonlinear tasks such as moving the system from a passively stable to a passively unstable controller-stabilized equilibrium. Our approach demonstrates the effectiveness of machine learning control and linear optimal control techniques for providing high-performance controllers in this challenging domain. The generality and transferability of the techniques to other systems will also be discussed.

4:49PM EJ.00004 Shape Optimization of Micro-Magnetic Locomotors, ERIC KEAVENY, SHAWN WALKER\textsuperscript{2}. MICHAEL SHELLEY, New York University, Courant Institute — Locomotion at the micro-scale is important in biology and in industrial applications such as targeted drug delivery and micro-fluidics. We present results on the optimal shape of a rigid body locomoting in 3-D Stokes flow. The actuation consists of applying a fixed moment and constraining the body to only move along the moment axis; this models the effect of an external magnetic torque on an object made of magnetically susceptible material. The shape of the object is parametrized by a 3-D centerline with a given cross-sectional shape. No a priori assumption is made on the centerline. Thus, we pose an infinite dimensional optimization problem and solve it with Boundary Integral and Variational methods. Sensitivities of the cost and constraints are computed variationally via shape differential calculus and a boundary integral formulation yields the boundary stresses. The optimization method can be considered as a sequential quadratic programming (SQP) approach. We report examples of locomotor shapes with and without different fixed payload/cargo shapes.

\textsuperscript{2}Presenter

5:02PM EJ.00005 Local flow control for active building facades, SRIKAR KALIGOTLA, WAYNE CHEN, MARK GLAUSER, Syracuse University — Existing facade design practices are for a passive and an impermeable shell to prevent migration of outdoor air into the building and to control heat transfers between the exterior environment and the building interior. An active facade that can respond in real time to changing environmental conditions like wind speed and direction, pollution load, temperature, humidity and light can lower energy use and maximize occupant comfort. With an increased awareness of cost and environmental effects of energy use, cross or natural ventilation has become an attractive method to lower energy use. Separated flow regions around such buildings are undesirable due to high concentration of pollutants, especially if the vents or dynamic windows for cross ventilation are situated in these regions. Outside pollutant load redistribution through vents can be regulated via flow separation control to minimize transport of pollutants.

5:15PM EJ.00006 A Closed-loop Suction Flow Control Study over a Pitching Turret, RYAN WALLACE, PATRICK SHEA, Syracuse University, VAITHIYANATHAN THIRUNAVUKKARASU, Clear Science Corp, RYAN SCHMIT, AFRL/RBTF, HAL CARLSON, Clear Science Corp, MARK GLAUSER, Syracuse University — Active flow control was implemented over a dynamically pitching hemispherical turret with a flat aperture in order to reduce the amount of turbulent fluctuations within the wake region. In this study unsteady suction was utilized as the control input for both open loop and closed loop control cases. The experiments were performed at the Subsonic Research Laboratory wind tunnel at Wright-Patterson Air Force Base at a high Reynolds number flow in which compressibility effects are present. It was clearly demonstrated with the open loop control cases that the suction had enough control authority to effect the baseline flow. The closed-loop control cases explored the effects of various low dimensional feedback applications, utilizing measurement-based estimation and regulators to control either fluctuating transport or the mean velocity. The ultimate goal of the closed-loop control cases was to observe a reduction in turbulent fluctuations in the wake while reducing the amount of control input.

5:28PM EJ.00007 Feedback control of wall turbulence for drag reduction with finite spatial and temporal resolution, BETTINA FROHNAPEL, ALEXANDER STROH, TU Darmstadt, YOSUKE HASEGAWA, NOBUHIDE KASAGI, The University of Tokyo — Among various active control strategies, feedback control generally offers better control performance with smaller power consumption and better deal of structure. Abundant evidence from nature demonstrates that this structure can be exploited to achieve high performance at dynamical tasks. In this talk, we present experimental work on a simple fluid-body system, a hydrodynamic analogue to the well-studied “cart-pole” system. Examples include balancing an immersed robot at a passively unstable equilibrium, and more fundamentally nonlinear tasks such as moving the system from a passively stable to a passively unstable controller-stabilized equilibrium. Our approach demonstrates the effectiveness of machine learning control and linear optimal control techniques for providing high-performance controllers in this challenging domain. The generality and transferability of the techniques to other systems will also be discussed.

5:41PM EJ.00008 Optimization for the Design of Odd Feature Shapes, TREVOR MILLS, RYAN WALLACE, PATRICK SHEA, SRIKAR KALIGOTLA, WAYNE CHEN, MARK GLAUSER, Syracuse University — The goal of the optimization is to achieve better performance with less material usage. To achieve this, we propose an optimization method that can be considered as a sequential quadratic programming (SQP) approach. We report examples of locomotor shapes with and without different fixed payload/cargo shapes.
5:41PM EJ.00008 Direct simulation of a separated boundary layer under the influence of large-scale forcing.\(^1\) \(^2\) \(^3\) \(^4\) AYSE G. GUNGOR, MARK P. SIMENS, JAVIER JIMÉNEZ, U. Politecnica Madrid — The effect of large-scale forcing mimicking incoming wakes on a separated turbulent boundary layer over a flat plate is investigated by direct numerical simulation. The flow separates due to a strong adverse pressure gradient induced by suction along the upper simulation boundary, and the forcing Strouhal number \(St = fl/\dot{U}_g\) ranges from 0.25 to 2.9. The forcing, in which all the turbulent fluctuations except for the mean velocity defect are neglected, triggers the transition of the separated shear layer, and modifies the separated region. Each forcing pulse generates three roll-up vortices, which originate near the separation point and convect with approximately half the local free-stream velocity. The separation and reattachment points vary with the forcing frequency, but no other significant variations of the mean boundary layer properties are observed unless the separation bubble is allowed to fully reform. The separation lengths of the periodic cases can be estimated from a single recovery experiment in which the forcing is suddenly removed.

\(^1\) Funded by CICYT and CONSOLIDER

5:54PM EJ.00009 Control of vortex induced vibrations by suction and blowing. K. MURALIDHARAN, B.S.V. PATNAIK, IIT Madras, Chennai - 600 036 — Kármán vortex shedding behind bluff bodies is of interest in a wide range of technological applications. Vortex shedding past a stationary D-cylinder is controlled in our earlier investigations \([1]\). However, a flexibly mounted circular cylinder gives rise to vortex induced vibrations. The control of these vibrations is of both fundamental and practical interest as fluid submerged structures need suppression of vortex induced oscillations. Flow past a circular cylinder is numerically simulated by coupling mass, momentum conservation equations along with dynamical equations for the structure. An active flow control strategy based on suction and blowing is designed and implemented to assess the efficacy of this control methodology. This is achieved by suitably located suction and blowing on the cylinder surface. These actuators are designed such that, the suction and blowing together results in zero mass injection. This system is found to effectively annihilate the vortex induced vibrations, when the quantum of actuations is about thrice the free stream velocity. The blowing slot is located on the leeward side of the cylinder, while the suction slots are positioned at an angle of 100\(^\circ\) to the flow direction. The convective instability region is reduced, while the length of the wake formation region behind the body is controlled, with an attendant annihilation of the wake vortices. \([1]\) Patnaik BSV, Wei GW, Phy. Rev. Lett., 88, 054502, (2002).

6:07PM EJ.00010 Dissimilart control of momentum and heat transfer in wall turbulence with traveling wave-like wall blowing and suction. YOSUKE HASEGAWA, Center of Smart Interfaces, TU Darmstadt, NOBUHIDE KASAGI, Dept. Mech. Eng., The Univ. Tokyo, CENTER OF SMART INTERFACE, TU DARMSTADT COLLABORATION, DEPT. MECH. ENG., THE UNIV. TOKYO COLLABORATION — Because of the importance of fundamental knowledge on turbulent heat transfer for further decreasing entropy production and improving efficiency in various thermo-fluid systems, we revisit a classical issue whether enhancing heat transfer is possible with skin friction reduced or at least not increased as much as heat transfer. The answer that numerous previous studies suggest is quite pessimistic because the analogy concept of momentum and heat transport holds well in a wide range of flows. In the present study, we introduce the suboptimal control theory for achieving dissimilart control in one of the most canonical thermo-fluid system, namely, turbulent wall flow with control input a smooth and straight channel. The Fréchet differentials obtained clearly show that the responses of velocity and temperature fields to a given control input are quite different due to the fact that the velocity is a divergence-free vector while the temperature is a conservative scalar. By exploiting this inherent difference, the dissimilart control can be achieved even in flows where the averaged momentum and heat transport equations have the same form.

Sunday, November 21, 2010 4:10PM - 6:20PM — Session EK Acoustics I: Shear Flows Long Beach Convention Center 201B

4:10PM EK.00001 Investigation of the Near-Field Acoustic Properties of Supersonic Jets with Fluidic Enhanced Chevrons using Large-Eddy Simulations\(^1\) \(^2\) \(^3\) \(^4\) JUNHUI LIU, K. KAILASANATH, RAVI RAMAMURTI, Naval Research Lab, NICHOLAS HEEB, DAVID MUNDAY, EPHRAIM GUTMARK, University of Cincinnati — Since it has been found that chevrons reduce noise more effectively in the underexpanded operating range, but fluidic injection with constant injection mass flow rate is more effective in the overexpanded range, fluidic enhanced chevrons (a combination of chevrons and fluidic injection) are investigated numerically based on a MILES (Monotonically Integrated Large Eddy Simulations) approach. Both overexpanded and underexpanded jet conditions are tested and results are compared with experimental data. The mean flow field and the near-field noise spectra are also compared with those from the case with fluidic injection alone and the case with chevrons alone. It is found that fluidic enhanced chevrons have a larger impact in the overexpanded operating range than that in the underexpanded range.

\(^1\) Work sponsored by NRL 6.1 project.

4:23PM EK.00002 Non-linear parabolized stability equation (NPSE) models for predicting large-scale mixing noise of turbulent round jets\(^1\) \(^2\) \(^3\) ARNAB SAMANTA, Caltech, KRISTJAN GUDMUNDSSON, University of Twente, TIM COLONIUS, Caltech — We study sound generation from lower-frequency, large-scale wavepacket structures of turbulent round jets using PSE models. The computations use a set of subsonic and supersonic mean flows for which databases from PIV measurements and LES simulations, respectively, are available. Linear PSE models have previously shown good agreements with the amplitude and phase of microphone array data measured just outside the jet shear layer. Non-linear effects are likely to be important for the lower-order modes, near and beyond the closing of the jet potential core, where the wave amplitudes reach their maximum values. Unlike the LPSE evolution, which is independent of the initial amplitudes, an accurate estimate of the near-nozzle oscillations is necessary as the initial condition for non-linear PSE model simulation. This is obtained from experimental data or high-fidelity simulations. Studies show the non-linear evolution to be sensitive not only to the initial modal amplitudes but also to their phases, the number of modes retained in the solution and also any spurious noise that might be present in the mean flow measurements.

\(^1\) Support from NAVAIR through TTC Tech., Inc. and Cascade, Inc.

4:36PM EK.00003 Adjoint-based minimization of the sound radiated by a Mach 1.3 turbulent jet. JONGNGLAE KIM, DANIEL BODONY, JONATHAN FREUND, University of Illinois at Urbana-Champaign — A control optimization using the adjoint of the perturbed and linearized Navier–Stokes equations is applied to a simulation of a Mach 1.3 turbulent jet to reduce its radiated sound. The solution of the adjoint system provides gradient information for a minimization algorithm to circumvent the flow complexity and reduce the sound directly. Comparisons between the loud and the perturbed-but-quiet versions of the same jet are examined to identify sound mechanisms. The overall algorithm is designed such that the control can be optimized with degrees of freedom comparable to that of the numerical discretization or with constraints on its spatial or temporal profiles to reflect hardware limitations. The large-eddy simulation of the uncontrolled, baseline jet is carried out in curvilinear coordinates using a non-dissipative high-order finite-difference. The far-field sound is computed using a Ffowcs Williams and Hawkings surface. Turbulence and far-field sound statistics agree with experimental data. An unconstrained optimal control reduces the sound cost functional by 17%. The far-field sound is reduced at all angles with a maximum reduction of 2.7dB in the peak radiation direction. Constraining the control in actuator-like zones shows a similar result. Optimizations are ongoing.
4:49PM EK.00004 Effect of roughness shape on rough-wall boundary-layer noise. QIN YANG, MENG WANG, University of Notre Dame — Turbulent boundary-layer noise induced by arrays of 10 \times 4 sparsely distributed roughness elements is investigated using Lighthill's theory with acoustic sources obtained from large-eddy simulation. Two types of roughness elements, hemispheres and cuboids with the same height \( h = 0.124 \delta (h^+ = 168) \), are considered. The acoustic formulation shows that each roughness element acts as compact in-plane dipole sources strengthened by their images in the wall. The acoustic characteristics are found to be strongly dependent on the roughness shape. For the hemispherical array, the dipole sources are mainly generated by the interaction of hemispheres with incoming turbulent eddies, spanwise dipoles are stronger than streamwise dipoles, and the leading row of cuboid elements, under certain conditions, is responsible for the disturbance. For cuboid elements, unsteady separation and reattachment around the front edges are important. The acoustic sources, the spanwise dipoles are slightly weaker than the streamwise dipoles, and the leading row produces the strongest radiation. Correlations and coherence between dipole sources associated with neighboring roughness elements are found to be weak, particularly for cuboids and in the spanwise direction.

5:02PM EK.00005 Identifying Lighthill source term with large-eddy simulation of subsonic turbulent jet. HYUNSUN LEE, ALI UZUN, MOHAMMAD YOUSUFF HUSSAINI, Florida State University, FLORIDA STATE UNIVERSITY TEAM — An acoustic analogy analysis based on the decomposition of the Lighthill source term into subterms is discussed in light of a high-fidelity large-eddy simulation of a subsonic turbulent jet from a baseline nozzle. These subterms consist of density, velocity, vorticity and dilatation fields, representing their reciprocal nonlinear interactions. To understand the aerodynamic noise generation mechanism, intrinsic links between turbulence and emitted sound waves, such as cross-correlation functions, are required. This causality method is directly adopted to the LES data to identify the fundamental noise sources. The cross-correlation between each spatial subterm in the near field and acoustic pressure fluctuation at a far field position is calculated, showing its contribution to noise generation. Three principal noise production terms, related to turbulence kinetic energy and Lamb vector, are witnessed and interpreted. The results show encouraging agreement with previous predictions. Future work will extend the observation to chevron nozzle jet. A comparison of the correlation profiles will possibly lead us to characterize the distinct structures of the chevron nozzle and baseline round nozzle jets. Furthermore, this study is expected to help in better understanding and assessing of noise control devices.

5:15PM EK.00006 Effects of heating and Mach number on global modes of high-speed jets. JOSEPH NICHOLS, SANJIVA LELE, Center for Turbulence Research, Stanford University — The noise produced by high-speed jets is analyzed by global mode decomposition over a range of Mach numbers and jet-to-ambient density ratios. A massively parallel shift-and-invert Arnoldi method is used to extract global modes from simulations of the fully compressible linearized Navier–Stokes equations. Both laminar and turbulent mean (satisfying the Reynolds averaged Navier-Stokes equations with the k-epsilon model of Tam and Thies) base flows are considered. The least-stable portions of the global spectra were observed to split into two types of branches, which are shown to be linked to Kelvin-Helmholtz and Tam-Hu instability waves, respectively. Because they are upstream propagating, Tam-Hu waves are neglected by traditional analyses based on the parabolized stability equations. Significant transient growth is recovered through an optimal superposition of the global modes, and it is found that non-normality increases with decreasing density ratio as well as with decreasing Mach number.

5:28PM EK.00007 Jet Noise Source Parameterization Based on Far-Field Sound Intensity Distributions. DIMITRI PAPAMOSCHOU, U.C. Irvine — Predictive tools for the interaction of jet noise with airframe surfaces require a model for the jet noise source that is simple yet physically relevant. The model used here consists of the incoherent superposition of a wavepacket and a monopole. Data to parameterize such model are limited primarily to far-field sound pressure level spectra. For a given frequency, the source parameters are determined by matching, in a least-squares sense, the polar intensity distribution in the far field. Even though only a small fraction of the wavepacket spectral content radiates to the far field, it is possible to construct models for the jet noise source that reproduce well the far-field polar intensity distribution and contain reasonable wavepacket parameters. In particular, the directivity pattern of the sound intensity provides strong guidance as to the azimuthal mode content of the wavepacket. The contribution of the monopole is small and typically on the order of 10% of the peak intensity. The model is extended to non-axisymmetric jets and to jets with limited azimuthal coherence.

5:41PM EK.00008 Acoustics and Mean Flow of Coaxial Jets with Variable Velocity and Area Ratios. SARA ROSTAMIMONEZI, DIMITRI PAPAMOSCHOU, U.C. Irvine — We investigate experimentally the far-field acoustics, noise source distributions, and mean flow structure of coaxial jets with secondary-to-primary velocity ratio \( U_s/U_p \) ranging from 0 to 1 and diameter ratios of 1.54, 1.64, and 1.98. The mean velocity field is characterized in terms of the length of the primary core \( L_p \), defined as the high-speed region of the jet, and the length of the secondary \( L_s \), defined by the outer inflectional points of the radial velocity profile. The ratio \( L_s/L_p \) increases with velocity ratio and with diameter ratio. For velocity ratios between 0.4 and 0.8, the elongation of the secondary core relative to the primary core is accompanied by suppression of high-frequency noise sources near the jet exit. This results in a reduction of far-field sound pressure levels, which is particularly strong when the results are scaled to constant thrust. Given the importance of \( L_s/L_p \), models for the core lengths are constructed and preliminary correlations for the reduction of overall sound pressure level versus \( L_s/L_p \) and other critical parameters are offered.

5:54PM EK.00009 The frequency spectrum of sound generated by turbulent shear flows. GUOWEI HE, XIN ZHAO, XING ZHANG, LNM, Institute of Mechanics, Chinese Academy of Science — The frequency spectrum of sound radiated by turbulent shear flows is analytically calculated from Lighthill’s acoustic analogy by evaluating space-time correlations. The turbulent shear flow is a simple model for jet noise, with the assumptions that its large scales are steady shear and its small scales are isotropic. Ribner (J. Fluid Mech. 38 1-24 1969) calculates the noise intensity for this model, which shows the basic directivity of jet noise. Recently, a non-frozen flow model is developed for space-time correlations in turbulent shear flows (Phys. Rev. E. 79 046316 2009) and experimentally verified against the Rayleigh-Benard convection (Phys. Rev. E. 81 065303 2010). We use the non-frozen flow model to calculate the noise spectra for turbulent shear flows. The result obtained is explicitly dependent on the lateral angle. It offers a scaling form of noise spectra at high frequencies, which is consistent with the two noise source model by Tam et. al. (AIAA paper 96-1716). The results are obtained as well as compared with the experiments and numerical simulations.
6:07PM EK.00010 Study of supersonic components in high-speed turbulent jets using wavenumber-frequency domain filtering and POD\textsuperscript{1} \textsuperscript{1}JAIYOUNG RYU, SANJIVA LELE, Stanford University — Near-field characteristics of supersonic components are investigated using LES database by Bodony and Lele (2005). Three unheated jets with jet Mach number ranging from 0.51 to 1.95, and one heated transonic jet are considered. Supersonic components are decomposed from full flow field using wavenumber-frequency domain filtering. Spatial structure of the fluctuating pressure field is obtained by computing proper orthogonal modes (POD modes) of the full and filtered data. POD modes of subsonic jet ($M_j = 0.51$) reveals large scale-disparity between full and supersonic components. For the supersonic jet at $M_j = 1.95$, the energetic structures of the pressure field also contribute significantly to the supersonic components and scale disparity is absent. The subsonic pressure variance can be rescaled by $V_j^2$ (i.e. $P_i = \lambda V_j^2$, which is an expected scaling for turbulence associated pressure fluctuations, whereas supersonic pressure variance can be rescaled with $V_j^2$, which is consistent with the far-field noise intensity scaling associated with Lighthill's analogy. Filtered velocity components are also rescaled and similar pattern is observed (i.e. supersonic near-field velocity scales as far-field disturbances).

\textsuperscript{1}This research has been supported by the Boeing Company.

Sunday, November 21, 2010 4:10PM - 6:20PM
Session EL Biofluids: Physiological Respiratory Long Beach Convention Center 202A

4:10PM EL.00001 Regional airflow and particle distribution in the lung with a 3D-1D coupled subject-specific boundary condition\textsuperscript{1} \textsuperscript{1}JWOOONG CHOI, YOUBING YIN, ERIC HOFFMAN, The University of Iowa, MERRYN TAWHAI, The University of Auckland, CHING-LONG LIN, The University of Iowa — Correct prediction of regional distribution of inhaled aerosols is vital to improve pulmonary medicine. Physiologically consistent regional ventilations of airflow and aerosol particles are simulated with a 3D-1D coupled subject-specific boundary condition (BC). In 3D CT-resolved 7-generation airways, large eddy simulations are performed to capture detailed airflow characteristics and Lagrangian particle simulations are carried to track the particle transport and deposition. Results are compared with two traditional outlet BCs: uniform velocity and uniform pressure. Proposed BC is eligible for physiologically consistent airflow distribution in the lung, while the others are not. The regional ventilation and deposition of particles reflect the regional ventilation of airflow. In this study, two traditional BCs yield up to 98% (334%) over-prediction in lobar particle ventilation (deposition) fraction. Upper to lower particle ventilation ratios of both left and right lungs read $\sim0.4$ with the proposed BC, while those for the other two BCs vary with the error up to 73%.

\textsuperscript{2}The work is sponsored by NIH Grants R01-ES-005823, R01-HL-064368, S10-RR-022421, and R01-HL-094315.

4:23PM EL.00002 Newtonian to non-Newtonian flow transition in lung surfactants, AMIR SADOUGHI, AMIR HIRS, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — The lining of normal lungs is covered by surfactants, because otherwise the surface tension of the aqueous layer is too large to allow breathing. Lack of functioning surfactants can lead to respiratory distress syndrome, on water surface and measure the interfacial velocity field. The measured interfacial velocity is compared to Navier-Stokes computations with the Boussinesq-Scriven surface model. Results show that DPPC monolayer behaves (i) purely elastically at low surface pressures on water, (ii) viscoelastically at modest surface pressures, exhibiting non-zero surface shear viscosity that is independent of the shear rate and flow inertia, and (iii) at surface pressures approaching film collapse, DPPC loses its fluid characteristics, and a Newtonian surface model no longer captures its hydrodynamics.

4:36PM EL.00003 Stability analysis of the pulmonary liquid bilayer, DAVID HALPERN, University of Alabama, JAMES GROTBERG, University of Michigan — The lung consists of liquid-lined compliant airways that convey air to and from the alveoli where gas exchange takes place. Because the airways are coated with a bilayer consisting of a mucus layer on top of a periciliary fluid layer, a surface tension instability can generate flows within the bilayer and induce the formation of liquid plugs that block the passage of air. This is a problem for example with premature neonates whose lungs do not produce sufficient quantities of surfactant and suffer from respiratory distress syndrome. To study this instability a system of coupled nonlinear evolution equations are derived using lubrication theory for the thicknesses of the two liquid layers which are assumed to be Newtonian. A normal mode analysis is used to investigate the initial growth of the disturbances, and reveals how the grow rate is affected by the ratio of viscosities $\lambda$, film thicknesses $\eta$ and surface tensions $\Delta$ of the two layers which can change by disease. Numerical solutions of the evolution equations show that there is a critical bilayer thickness $\eta_c$ above which closure occurs, and that a more viscous and thicker layer compared to the periciliary layer closes more slowly. However, $\eta_c$ is weakly dependent on $\lambda$, $\eta$ and $\Delta$. We also examine the potential impact of wall shear stress and normal stress on cell damage. This work is funded by NIH HL85156.

4:49PM EK.00004 Airflow Simulation and Particle Deposition in a 3D Rat Lung Model, JESSICA OAKES, Mechanical and Aerospace Engineering, Department of Medicine, Division of Physiology at University of California, San Diego, SHAWN SHADDEY, Department of Mechanical, Materials & Aerospace Engineering, Illinois Institute of Technology, CHANTAL DARQUENNE, Department of Medicine, Division of Physiology at University of California, San Diego, ALISON MARSDEN, Mechanical and Aerospace Engineering at University of California, San Diego — Knowledge of the fate of aerosols in the lung is needed to understand the efficiency of inhaled drug therapy. Invasive animal experiments and imaging allows for detailed quantitative comparison with computational modeling. In this study we built a three-dimensional (3D) airway tree model using rat magnetic resonance images. A custom 3D finite element solver was used to obtain animal specific velocities and pressures. Inlet boundary conditions were chosen to match a previous rat ventilation experiment and resistance outlet boundary conditions were selected to match a desired airflow split based on uniform ventilation. The Maxey-Riley particle equations were solved using Lagrangian particle tracking methods with realistic aerosol particle dimensions and density. The particle dynamics were validated using analytical solutions in idealized geometries. The impact of the choice of outlet boundary conditions for airflow simulations is quantified and aerosol particle deposition and distribution within the lung lobes are explored.

5:02PM EL.00005 Numerical investigation of aerosolized drug delivery in the human lungs under mechanical ventilator conditions, TIMOTHY VANRHEIN, ARINDAM BANERJEE, Missouri S&T — Particle deposition for aerosolized drug delivery in the human airways is heavily dependent upon flow conditions. Numerical modeling techniques have proven valuable for determining particle deposition characteristics under steady flow conditions. For the case of patients under mechanical ventilation, however, flow conditions change drastically and there is an increased importance to understand particle deposition characteristics. This study focuses on mechanically ventilated conditions in the upper tracheo-bronchial (TB) region of the human airways. Solution of the concomitant phase flow is done with a suitable turbulence model in conjunction with a realistic model of upper TB airways. A discrete phase Euler-Lagrange approach is applied to solve for particle deposition characteristics with a focus on the effect of the ventilator inlet waveform. The purpose of this study is to accurately model flow conditions in the upper TB airways under mechanically ventilated conditions with a focus on real-time patient specific targeted aerosolized drug delivery.
5:15PM EL.00006 Unsteady Simulation of a Human Respiratory System with Micron-Particles

SHAHAB TAHERIAN¹, CEERS/CSULB, HAMID RAHAI², MAE Dept., CEERS/California State University, Long Beach, TOM WADDINGTON³, VA Hospital, Long Beach, California — Unsteady numerical simulations of air flow, mixed with micron particles, through a human lung conducting zone during inhalation/exhalation process have been performed. The process included importing images from a high resolution MRI into a CFD software, generation of the CFD model and then CFD simulation over a 4 seconds cycle. The inlet diameter was 16 mm and the flow rate was 7 liter/min. The implicit-unsteady Reynolds Average Navier-Stokes equations with the Wilcox K-ω turbulence model were used for the simulation. The micron particles were solid round lead with 1000 Kg/m³ density. Results indicate high correlation between regions of the secondary flows and particle deposits. This was mostly evident in the main bronchus. While most particles exit the lung during the exhalation process, however, areas of re-circulating flow and near the walls continue to have some particle deposits.

¹Graduate Assistant
²Professor and Director
³Associate Faculty Member

5:28PM EL.00007 Particle Deposition During Airway Closure¹, CHENG-FENG TAI, University of Michigan, DAVID HALPERN, University of Alabama, JAMES GROTBERG, University of Michigan — Inhaled aerosol particles deposit in the lung and may be from environmental, toxic, or medical therapy sources. While much research focuses on inspiratory deposition, primarily at airway bifurcations due to inertial impaction, there are other mechanisms that allow the particles to reach the airway surface, such as gravitational settling and diffusion depending on particle size. We introduce a new mechanism not previously studied, i.e. aerosol deposition from airway closure. The airways are lined with a liquid layer. Due to the surface tension driven instability, a liquid plug can form from this layer which blocks the airway. This process of airway closure tends to occur toward the end of expiration. In this study, the efficiency of the impaction of the particles during airway closure will be investigated. The particles will be released from the upstream of the airway and convected by the air flow and deposited onto the closing liquid layer. We solve the governing equations using a finite volume approach in conjunction with a sharp interface method for the interfaces. Once the velocity field of the gas flow is obtained, the path of the particles will be calculated and the efficiency of the deposition can be estimated.

¹We acknowledge support from the National Institutes of Health grant number NIH HL85156.

5:41PM EL.00008 Pulsatile flow past a single oscillating cylinder, ROBINSON SEDA, ADNAN QAMAR, JOSEPH BULL, University of Michigan — The potential for oscillating fibers to modify flow within a new artificial lung design is first examined in the present fundamental fluid mechanics study of flow past a single oscillating cylinder. This new design is intended to provide better gas exchange through vorticity enhancement by oscillating microfibers (cyinders) in a pulsatile flow environment. The Keulegan-Carpenter number (KC=Uo/D<ω) was used to describe the frequency of the oscillating cylinder (∇c) while the pulsatile free stream velocity was fixed by imposing Uo/Kc=1 for all cases investigated. The parameters investigated in this study were amplitude of oscillation (0.5D<ω<3D), kc corresponding to 1<ω<3 and Reynolds number (5<Re<20), all equivalent to operating conditions of the TAL. Vorticity was enhanced up to 246% from the steady state condition for high amplitudes and low kc for all Re. An opposite trend was observed for the drag coefficient. A "lock-in" phenomenon (cylinder oscillating frequency matching the vortex shedding frequency) was found when kc=1 for all cases. A jump in the drag coefficient was observed and attributed to this operating regime. These results suggest that this new design of the TAL could potentially enhance gas exchange through oscillation of the microfibers with a decrease in the drag coefficient if operating far from the lock-in regime. This work was supported by NIH grants R01HL69420 and R01HL89043.

5:54PM EL.00009 Coughing and sneezing, LYDIA BOUROUIBA, ABIY TASISSA, JOHN BUSH, Massachusetts Institute of Technology — The emergence and explosive spread of virulent viral (e.g., H1N1, SARS) and bacterial (e.g., Tuberculosis) infections is a problem of global interest with enormous human and economic consequences. The nature of contact between infected and non-infected persons greatly influences the outcomes of the disease epidemic; nevertheless, the definition and mechanisms leading to contact remain nebulous. We here examine the manner in which fluid dynamics modeling can assist in our understanding of contact and transmission of respiratory diseases. Particular attention is given to modeling the effluent of discrete exhalation events (e.g., coughing, sneezing) as multiphase thermals, and to predicting the range of pathogen-bearing droplets.

6:07PM EL.00010 Numerical Simulations of the Propagation of a Liquid Plug through a 2D Airway Bifurcation¹, BENJAMIN L. VAUGHAN, JR., JAMES B. GROTBERG, University of Michigan — Numerous medical therapies require the instillation of liquids plugs and their delivery throughout the pulmonary airways. This process and the effect on the resulting liquid distribution is controlled by a number of parameters, including airway orientation with respect to gravity, initial plug volume, liquid physical properties, and the imposed airflow rate which drives the plug from behind. The airflow rate defines an operative Capillary number, Ca, and the influence of gravity appears as an effective Bond number, Bo, whose magnitude varies with orientation. In this study, we develop a numerical method for solving the propagation of a liquid plug into a two-dimensional airway bifurcation consisting of a parent channel branching into two daughter channels. We measure the splitting ratio, RS, which is defined as the ratio of the liquid plug volumes between the daughter branches. RS increases with Ca and asymptotes to 1 as Ca goes to infinity, which corresponds to an equal split, while increasing Bo requires a higher value of Ca for an equal split. We also examine the normal and shear stresses on the bifurcation walls and observe that the stresses on the upper walls increase as Bo increases while the stresses on the lower walls decrease as Bo increases.

¹Supported by NIH grants HL85156 and HL84370.

Sunday, November 21, 2010 4:10PM - 6:20PM –
Session EM Microfluids: General III: Microchannels

4:10PM EM.00001 Taylor-Aris dispersion in time-dependent laminar channel flows, SØREN VEDEL HENRIK BRUUS, Department of Micro- and Nanotechnology, Technical University of Denmark — The effective axial diffusion of solute concentrations advected in channel flows is known as Taylor-Aris dispersion [1,2]. Due to the no-slip condition, particles near the walls are displaced less than those close to the channel center axis, leading to concentration gradient perpendicular to the axis and an enhanced axial diffusivity. In many applications the velocity field is unsteady, but concentration dispersion in such time-dependent flows is largely unexplored, except for transient dispersion of an initial concentration profile in a steady flow [3], and dispersion in a velocity field with one harmonically oscillating component superimposed on a steady component [4]. We present a mathematical theory for Taylor-Aris dispersion in a straight channel with an arbitrary time-dependent flow, based on Fourier expansion of the velocity field, valid for all times and all values of the Péclet number. The theory is applied to different time-dependent flows in channels of different cross sections, and we discuss the new phenomena arising by adding an increasing number of higher harmonics.

4:23PM EM.00002 Designing patterned microchannels to separate colloid-polymer suspensions\(^1\). HASSAN MASoud, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using dissipative particle dynamics, we examine the flow of colloid suspensions in microfluidic channels with patterned walls. The distribution of colloids in a channel is set by the competition between diffusion and hydrodynamic effects. We show that the distribution can be altered by introducing tilted nanoscopic posts protruding from internal walls of a microchannel. Specifically, we demonstrate that depending on the post orientation, the patterned walls can either hydrodynamically attract nanoscale objects suspended in the flowing fluid or prevent their depositions by repelling them away from solid walls. Furthermore, surfaces decorated with tilted posts can discriminate nanoscopic entities with regard to their shape and, thus, can be utilized for separating colloid-polymer mixtures.

\(^1\)Financial support from the ACS-Petroleum Research Fund is gratefully acknowledged.

4:36PM EM.00003 The Effect of Aspect Ratio on Taylor Dispersion in Oscillatory Poiseuille Flow in Rectangular Channels, JINKEE LEE, ELEJDIS KULLA, ANUBHAV TRIPATHI, Brown University, ANUJ CHAUHAN, University of Florida — The presence of size walls is known to lead to significant increase in dispersion in uniaxial Poiseuille flows even for very large aspect ratios. This presentation focuses on exploring the effect of the side walls on dispersion in oscillatory Poiseuille flows in rectangular channels. The method of multiple time scales with regular expansions is utilized to obtain analytical expressions for the effective dispersivity \(D^e\). The Navier-Stokes slip-flow solution is also used to elucidate a singular limit reported in the literature for oscillatory heating of a gas layer or to layers of arbitrary size with heating time-scales that are shorter than the mean collision time. Our analytical results are complemented by low-variance simulations. The effective dispersivity is of the form \(D^e_{1D} = Pe^f \Omega, Sc, \chi\) where its dependency on the dimensionless oscillating frequency \(\Omega\), the Schmidt number and the aspect ratio \(\chi\) of the channels is non-explicit. The effect of various parameters on dispersion coefficient is explored numerically and also through asymptotic expressions that are valid in some limiting cases. For small \(\Omega\) the dispersion coefficient for the oscillatory flow approaches the time-averaged dispersion of the unidirectional Poiseuille flow and for large \(\Omega\), \(D^e_{1D}\) scales as \(Pe^f/\Omega^2\). We believe that the results of this study will enhance our understanding of transport in microscale systems that are subjected to oscillating flows.

4:40PM EM.00004 Dynamics of pulsatile flows through elastic microtubes, OMER SAN, ANNE STAPLES, Virginia Tech — We investigate pressure driven transient flows of incompressible Newtonian fluids through circular microtubes with thin elastic walls under the long-wavelength and small deformation assumptions. An analytical solution of the coupled fluid and solid equations is found using the Navier slip boundary condition and is shown to include some existing Womersley solutions as limiting cases. The effect of the slip length at the fluid-solid interface of the flexible microtube is analyzed for oscillatory pressure gradients using a range of slip-ratio and frequency parameters. We find that for a steady pressure gradient, slip at the boundary simply adds a translational velocity and does not lead to material deformations, while for pulsatile flows with oscillating pressure gradients, the influence of the slip length becomes nonlinear and affects the flow rate, velocity profile, and shear stress. We compare the solutions for elastic and rigid walls with and without slip boundary conditions for broad ranges of the relevant parameters. We show that the elasticity of the microtube couples nonlinearly with the slip velocity and can greatly enhance the flow rate, significantly changing its maximum value and effective range as a function of Womersley number, compared to the no-slip case. Additionally, we find that increasing the slip length produces less shear stress, which is consistent with the nearly frictionless interfaces observed in many microscale experiments.

5:02PM EM.00005 An examination of collisional Lattice Boltzmann Method for microchannel flows, BOE GREEN, PRAKASH VEDULA, University of Oklahoma — A new computational approach for prediction of microchannel flows, which accounts for the full collision operator of the Boltzmann equation via a lattice framework, is presented. Unlike the widely used Lattice Boltzmann Method (LBM), our approach, called the collisional Lattice Boltzmann Method (cLBM), does not make any a priori assumptions on the equilibrium state and hence is capable of handling general nonequilibrium flows (i.e. over a wide range of Knudsen numbers). In cLBM, an operator splitting approach is used for solution of the Boltzmann equation, where representative populations of notional particles are streamed along the underlying lattice from all lattice nodes and the effects of collisional relaxation at each node are accounted for via a solution of a system of differential equations, derived from the full collision operator. This approach not only preserves several symmetries of the full collision operator, but is also structured to account for the evolution of selected generalized moments of the distribution (including conservation of mass, momentum, energy). Simulations of microchannel Couette and Poiseuille flows (including pressure driven and body force driven cases) over a broad range of Knudsen numbers, using a D3Q7 lattice structure, show that the results obtained from cLBM are in good agreement with those obtained from conventional LBM (relying on equilibrium based BGK model).

5:15PM EM.00006 Droplet and Slug Detachment and Entrainment in Microchannel Gas Flows, BRIAN ROBINSON, BRIAN CARROLL, CARLOS HIDROVO, The University of Texas at Austin — Liquid droplet and slug dynamics in a confined microchannel high speed gas flow is an important phenomenon with applications in two-phase micromixers, spray cooling for point source heat rejection, and water management in proton exchange membrane fuel cells. Thus, the ability to understand, predict, and control droplet growth, detachment, entrainment, and possible breakup is crucial. When subjected to a gas flow in a standard T-junction arrangement, experimental studies have shown that droplet and slug detached characteristics are determined by the gas Reynolds number, site geometry, and liquid/solid interfacial tension. Increasing the gas Reynolds number reduces the volume of the detached droplets and slugs while injection geometry and interfacial tension influence droplet and slug tail growth and formation of liquid films. Additionally, droplets can grow and detach with and without contact with adjacent channel walls due to site contaminants, geometry imperfections, and surface treatments, thereby adding complexity to the detachment process.

5:28PM EM.00007 Gas-flow animation by unsteady boundary heating in a microchannel\(^1\). NICOLAS HADJICONSTANTINOU, Mechanical Engineering Department, MIT, GREGG RADTKE, Mechanical Engineering Department, MIT, AVSHALOM MANELA, Aerospace Engineering Department, Technion — We study the response of a one-dimensional gas layer due to unsteady boundary heating. Analytical results are presented for the slip-flow/Navier-Stokes and collisionless limits. The latter is applicable to gas layers that are thinner than the molecular mean free path or to layers of arbitrary size with heating time-scales that are shorter than the mean collision time. Our analytical results are complemented by low-variance simulations of the Boltzmann equation, which are useful for establishing the limits of validity of the closed-form predictions, as well as bridging the gap between them. In particular, we consider the gas response to step-jump heating and show that the slip-flow solution captures the correct gas behavior for times as short as few collision times. The Navier-Stokes slip-flow solution is also used to elucidate a singular limit reported in the literature for oscillatory heating of a dynamically incompressible fluid.

\(^1\)This work was supported, in part, by the Singapore-MIT Alliance.

5:41PM EM.00008 ABSTRACT WITHDRAWN
5:54PM EM.00009 Effect of channel turn on the trajectory of an electrophoretic particle, DUSTIN HOUSE, HAOXIANG LUO, Vanderbilt University — Streamlines of non-particle-laden flow are often used as a convenient method to predict the trajectory of particles driven through a microchannel by electrophoresis. However, the validity of this approach it is not clear when the channel geometry is complex and when the particle size is large compared to the characteristic length scale of the channel. To address this issue, we have developed an accurate numerical approach based on the boundary-element method to solve the coupled electric field, flow and particle motion. From this, we simulate a spherical particle moving in a bent cylindrical channel. In the simulation, both the particle and channel walls are non-conducting, and the electrical double layers adjacent to the solid surfaces are assumed to be thin with respect to the particle radius and to the particle-wall gap. The result shows that the particle trajectory deviates from the flow streamlines (in the absence of the particle) when the turning radius is small and the particle is close to the inner side of the turn. The effect of the particle-to-cylinder size ratio will be also be presented.

6:07PM EM.00010 Effect of surface condition on the flow in segmented gas-liquid microreactors1, SHAHRAH POUYA, MANOOCHEHR KOCHESFAHANI, Michigan State University — The mixing process within segmented gas-liquid microreactors is of significant importance in design and optimization of devices for high throughput material synthesis. In a typical slug flow regime the liquid slugs are connected through a thin liquid film that plays an important role in hydrodynamics of the microreactor flow. Among the parameters that can influence the thin film layer, and the overall flow, is the surface condition of microchannel walls. We present preliminary results of this influence in the segmented gas-liquid flow of Ethanol/Nitrogen within PDMS microreactors. The results are presented specifically for microreactors with different level of roughness on the channel walls. The range of stable slug flow regime and behavior of liquid film are studied as a function of surface roughness.

1This work was supported by the CRC Program of the National Science Foundation, Grant Number CHE-0714028.

Sunday, November 21, 2010 4:10PM - 6:20PM –
Session EN Vortex Flows: Vortex Rings Long Beach Convention Center 202C

4:10PM EN.00001 The stability of a family of vortex rings1, CLARA O’FARRELL, JOHN O. DABIURI, California Institute of Technology — Jetting swimmers, such as squid or jellyfish, propel themselves by forming axisymmetric vortex rings. In order to evaluate the performance of these swimmers, we must assess the optimality of the vortex wakes they produce, which requires an understanding of their stability. We consider the Norbury family of vortex1 as a model for the vortex rings produced by jetting swimmers. Pozrikidis4 has studied the stability of Hill’s spherical vortex under axisymmetric prolate and oblate shape perturbations. However, the stability of other members of the Norbury family to axisymmetric perturbations of the type that might occur during the vortex formation process in jetting swimmers is unknown. In order to assess the stability of different members of the family, we introduce physically pertinent shape perturbations and simulate their development in a manner akin to Pozrikidis’ analysis.

1Funded by the NSF Fluid Dynamics (CBET-0754493) and Biological Oceanography (OCE-0623475) Programs.

4:23PM EN.00002 Motion of a vortex ring with swirl1, MING CHENG, JING LOU, Institute of High Performance Computing, TEE TAI LIM, National University of Singapore, IHPC-NUS COLLABORATION — Motion of vortex rings has been subject of theoretical and experimental studies since the time of Lord Kelvin simply because of its fundamental significance in flow physics and its practical importance in engineering applications. In this paper, we use a lattice Boltzmann method to simulate the motion of a vortex ring with and without swirl in a viscous incompressible fluid. We study the effect of swirl on the dynamics of an isolated three-dimensional vortex ring at a Reynolds number of 800. Our results show that the evolution of the vortex ring is affected by both the magnitude of swirl and the vortex core size. Increasing swirl for a fixed core size causes vortex ring to slow down or even travel backward initially. Increasing core size not only reduces the propagation speed of the ring, it also increases the duration of backward motion when the swirl is sufficient high. Moreover, it is found that while a weak swirl causes vortex filaments to undergo helical winding, a sufficiently strong swirl transforms these windings into convoluted three-dimensional vortex structure with vortex loops trailing behind it. Each of these vortex loops may reconnect with itself, through the process of vortex reconnection, to form a ringlet.

4:36PM EN.00003 Effect of a ground plane on a turbulent vortex ring trajectory, MARIA-LAURA BENINATI, Bucknell University, MICHAEL MCRERLEAN, MICHAEL KRANE, ARNOLD FONTAINE, Pennsylvania State University — Experiments were conducted to assess how a turbulent (Re=20000) vortex ring’s trajectory is affected by a ground plane parallel to its initial trajectory. This study, part of a larger effort in vortex-particle interaction, aims to characterize the vortex ring flow disturbance that interacts with a particle. Vortex ring motion was characterized for four distances between the initial vortex ring axis and the ground plane. Characterization included vortex centroid motion and diameter from high-speed video, vortex ring circulation from DPIV, and the wall pressure disturbance time traces. It was observed that in all cases the vortex ring trajectory is deflected toward the plane, ending in a collision. As plate height is decreased, the collision occurs closer to the ring generator, the wall pressure signature is also more intense, and the symmetry of the ring is affected more strongly.

4:49PM EN.00004 Convection of a Vortex Ring Parallel to a Plane Wall1, VANORA O’LOUGHLIN, DOUG BOHL, Clarkson University — In this work we investigate the motion and structure of a vortex ring convecting in a quiescent fluid parallel to a plane wall. The vortex rings were visualized using Laser Induced Fluorescence and recorded digitally. The plane wall was placed between 0.4-1.7 ring diameters away from the center of the ring. The results show that the vortex ring trajectory diverted towards the wall. The initial trajectory was described by inviscid flow models. As the ring came closer to the wall the interaction became viscous in nature. The portion of the ring closest to the wall interacted with the wall first and lost its coherence. The upper portion of the ring continued to convect towards the wall. This region induced a wall boundary layer that eventually separated and orbited the primary region of vorticity. In some cases the primary vortex ring also rebounded from the wall. The interaction was qualitatively similar to that of a vortex ring/oblique wall interaction once the trajectory was diverted toward the wall.

1This work supported by NSF Grant 0845882.

5:02PM EN.00005 Vortex Ring Dynamics in Radially Confined Domains, KELLEY STEWART, CASANDRA NIEBEL, SUNGHWAN JUNG, PAULOS VLACHOS, Virginia Tech — Vortex ring dynamics have been studied extensively in semi-infinite quiescent volumes. However, very little is known about vortex-ring formation in wall-bounded domains where vortex wall interaction will affect both the vortex ring pinch-off and propagation velocities. This study addresses this limitation and studies vortex formation in radially confined domains to analyze the affect of vortex-ring wall interaction on the formation and propagation of the vortex ring. Vortex rings were produced using a pneumatically driven piston cylinder arrangement and were ejected into a long cylindrical tube which defined the confined downstream domain. A range of confinement domains were studied with varying confinement diameters. Velocity field measurements were performed using planar Time Resolved Digital Particle Image Velocimetry (TRDPIV) and were processed using an in-house developed cross-correlation PIV algorithm. The experimental analysis was used to facilitate the development of a theoretical model to predict the variations in vortex ring circulation over time within confined domains.
5:15PM EN.00006 Interaction of a Vortex Ring with Surfaces of Constant Porosity1, JOHN HRYNUK, DOUG BOHL, Clarkson University — The interaction of vortices with surfaces is a fundamental process in many natural and technological flow fields. In this work we study the interaction of a vortex ring with porous surfaces using Laser Induced Fluorescence. The surfaces studied were stainless steel screens with a constant open area of 65% but different wire diameters (0.017-0.267 cm). Three distinct interactions were observed: 1) For small wire diameters the vortex rings passed through the screen and maintained their coherence and size but with a much slower convection speed. Secondary rings were formed on the upstream side of the screen and convected back upstream; 2) For medium gage wires the vortex ring broke up as it passed through the screen but reformed into a coherent vortex later downstream; 3) For large gage screens the vortices broke up and did not reform downstream. The transition between the interaction types appeared to be dependent on shedding from the screen wires. Specifically, for the small gage screens no shedding from the screen was observed. The medium gage wire showed the beginning of vortex shedding off of the screen wires while the large gage wires showed clearly formed vortices being shed from the screen wires.

1This work supported by NSF Grant 0845882.

5:28PM EN.00007 Vortex rings from Sphagnum moss capsules, DWIGHT WHITAKER, SAM STRASSMAN, JUNG CHA, EMILY CHANG, XINYI GUO, Pomona College, JOAN EDWARDS, Williams College — The capsules of Sphagnum moss use vortex rings to disperse spores to suitable habitats many kilometers away. Vortex rings are created by the sudden release of pressurized air when the capsule ruptures, and are an efficient way to carry the small spores with low terminal velocities to heights where they can be carried by turbulent wind currents. We will present our computational model of these explosions, which are carried out using a 2-D large eddy simulation (LES) on FLUENT. Our simulations can reproduce the observed motion of the spore clouds observed from moss capsules with high-speed videos, and we will discuss the roles of bursting pressure, cap mass, and capsule morphology on the formation and quality of vortex rings created by this plant.

5:41PM EN.00008 Formation number of positively and negatively buoyant vortex rings, JAVIER RODRÍGUEZ-RODRÍGUEZ, CAROLINA MARUGÁN-CRUZ, Carlos III University of Madrid, CARLOS MARTÍNEZ-BAZÁN, University of Jaen — The formation process of both negatively and positively buoyant vortex rings in a piston/cylinder arrangement is investigated numerically with the aim of understanding the effect of buoyancy, characterized by a Richardson number, on the formation number. More specifically, the study focuses on how vorticity is distributed inside the vortex ring and how this vorticity distribution compares with the neutrally buoyant case. It is well known that the kinetic energy of a neutrally buoyant vortex ring, when made dimensionless with its impulse and circulation, has a universal value of $E_{n1}/3$. The limits of validity of this value for moderate Richardson numbers, both in the positively and negatively buoyant cases, are examined.

5:54PM EN.00009 Vortex ring refraction at large Froude numbers1, KERRY KUEHN, MATTHEW MOELLER, MICHAEL SCHULZ, DANIEL SANFELIPPO, Wisconsin Lutheran College — We have experimentally studied the impact of an initially planar axisymmetric vortex ring, incident at an oblique angle, upon a gravity-induced interface separating two fluids of differing densities. After impact, the vortex ring was found to exhibit a variety of subsequent trajectories, which we organize according to both the incidence angle, $\theta_i$, and the interface strength, defined as the ratio of the Atwood and Froude numbers, $A/F$. For grazing incidence angles ($\theta_i > 70$ deg.) vortices either penetrate or reflect from the interface, depending on whether the interface is weak or strong. In some cases, reflected vortices execute damped oscillations before finally disintegrating. For smaller incidence angles ($\theta_i < 70$ deg.) vortices penetrate the interface. When there is a strong interface, these vortices are observed to curve back up toward the interface. When there is a weak interface, these vortices are observed to refract downward, away from the interface. The critical interface strength below which vortex ring refraction is observed is given by $\log_{10}(A/F) = -2.38 \pm 0.05$.

1Work supported by a grant from the Research Infrastructure Program of the NASA Wisconsin Space Grant Consortium.

6:07PM EN.00010 An experimental study on merging of two co-axial co-rotating vortex rings, JAGANNADHA SATTI, University of Alaska Fairbanks, JIFENG PENG, University of Alaska Fairbanks — The merging of two co-axial, co-rotating vortex rings is studied experimentally. Two laminar vortex rings were generated consecutively from a piston-cylinder apparatus. The two rings propagate in the same direction and the spatial separation between them decreases until they start merging. Special cases of leapfrogging were also observed. Digital particle image velocimetry was used to measure the flow fields. Core sizes, trajectories and circulations were measured for individual rings before the merging, as well as afterwards for the merged ring. At low Reynolds number, the total circulation in the flow is relatively a constant before and after merging. However, at high Reynolds number, the total circulation decreases quickly upon the contact of two vortex ring cores, indicating the transition to a turbulent vortex ring during merging. The circulation of the merged ring later stabilizes at a less level, indicating the merged ring becomes laminar again after shedding some circulation. Comparison between results from this experimental study and previous theoretical and computational studies in the literature are also discussed.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session EP Nanofluids III Long Beach Convention Center 203A

4:10PM EP.00001 Atomistic-continuum hybrid simulations for nano-scale flows, PANDURANG KULKARNI, Department of Chemical Engineering, University of California at Santa Barbara, GAURAV TOMAR, Department of Mechanical Engineering, Indian Institute of Science, CHIA-CHUN FU, M. SCOTT SHELL, L. GARY LEAL, Department of Chemical Engineering, University of California at Santa Barbara — It is known that the continuum assumption breaks down when the length scale of a flow approaches few nanometers. Examples include dynamics of thin films and interfaces, slippage in nanochannels and complex biological flows. In this work we develop a hybrid multiscale model, which combines atomistic description in a spatially localized region with continuum description in larger part of the flow domain. The atomistic region is simulated using standard molecular dynamics (MD) with particles interacting via Lennard-Jones pair potential. The continuum part of the problem is solved using the boundary-integral method. The spatio-temporal coupling between the two descriptions is achieved through constrained dynamics in the overlap region. The proposed model is validated by simulating shear flows in channels. A quantitative agreement is found between the computed flow fields and the analytical solutions. The boundary-integral based continuum solver offers improved efficiency and stability over conventional CFD methods. The potential applications of the method in emerging nano-fluidics are discussed.
4:23PM EP.00002 Molecular dynamics simulation of non-Newtonian phenomena and shear-induced structural changes in atomic fluids. XIN YONG, LUCY ZHANG, Rensselaer Polytechnic Institute — The rheology and microstructure of dense simple fluids have been mysterious subjects for decades due to the lack of direct experimental investigation. In this study, we present a planar Couette flow using molecular dynamics simulations to investigate the properties such as the velocity profile, density distribution, temperature profile and fluid structure within a wide range of wall velocities or shear rates. Here, we examine both boundary-driven shear (with walls) and homogeneous shear (without walls). In the boundary-driven shear, we model a fluid slab confined between two smooth and rigid solid walls. The upper wall is assigned a velocity to induce a planar Couette flow. In the homogeneous shear, only fluid atoms are modeled. The shear flow is generated from a canonical ensemble with a superimposed linear velocity profile associated with the Lees-Edwards periodic boundary conditions. Solid-like fluid layers were observed in the boundary-driven shear. In the homogeneous shear, the string phase is formed at high shear rates, which results in dramatic shear thinning. At lower shear rates, crystallization of fluid atoms induced by a large-scale secondary flow may appear. The physical features of the fluid structures and the corresponding viscosities are compared in the two models.

4:36PM EP.00003 A Long-Range Electric Field Solver for Molecular Dynamics of Fluid-Solid Interfaces Based on Atomistic-to-Continuum Modeling. JEREMY TEMPLETON, REESE JONES, JONATHAN ZIM- MERMAN, BRYAN WONG, Sandia National Laboratories, Livermore CA; JONATHAN LEE, Rice University — Understanding charge transport processes at a molecular level using computational techniques is currently hindered by a lack of appropriate models for incorporating anistropic electric fields, as occur at charged fluid/solid interfaces, in molecular dynamics (MD) simulations. In this work, we develop a model for including electric fields in MD using an atomistic-to-continuum framework. Our model represents the electric potential on a finite element mesh satisfying a Poisson equation with source terms determined by the distribution of the atomic charges. The method is verified using simulations where analytical solutions are known or comparisons can be made to existing techniques. A calculation of a salt water solution in a silicon nanochannel is performed to demonstrate the method in a target scientific application.

4:49PM EP.00004 A New Approach in Processing Atomistic Simulations. LEOPOLD GRINBERG, GEORGE KARNIADAKIS, Brown University — Computing an ensemble average in unsteady flow simulations performed with the Molecular Dynamics or coarse-grained versions, e.g. the Dissipative Particle Dynamics method, typically requires phase averaging of numerical solution over large number of time periods and realizations. For faster and more accurate processing we propose a new approach based on the window-Proper Orthogonal Decomposition (WPOD) methodology developed by Grinberg et. al. (ABME, vol. 37, 2009). WPOD helps to extract components of the velocity fields with high correlation time, i.e., ensemble average, in a hierarchical manner. The method is very robust and easy to implement. It leads to at least ten-fold computational savings and is appropriate for steady and unsteady non-periodic in time flows. We will review the new technique and present results of 3D numerical simulations of unsteady flows in microvessels.

5:02PM EP.00005 Multiscale analysis of structure of confined simple fluids. TARUN SANGHI, NARAYANA ALURU, University of Illinois at Urbana-Champaign — We discuss our recently proposed multiscale approach, an empirical potential based quasi-continuum theory (EQT), to predict the equilibrium structure of confined fluids across multiple length scales. In EQT, Nernst-Planck’s equation is used to obtain self-consistent concentration and potential profiles of the confined fluid. The robustness, accuracy and computational efficiency of the framework are demonstrated by obtaining concentration and potential profiles of several simple Lennard-Jones type fluids (non-polar, spherical molecules such as Methane, Oxygen, Argon) confined in slit like geometries and comparing the results with molecular dynamics (MD) simulations. The extension of the framework for confined polyatomic fluids (linear rigid chain like molecules such as Ethane and Carbon-dioxide) is also discussed.

5:15PM EP.00006 Origin of line tension for a Lennard-Jones nanodroplet. JOOST H. WEIJS, Physics of Fluids, University of Twente, Enschede, ANTONIN MARCHAND, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, ESPCI, Paris; JACCO H. SNOEIJER, Physics of Fluids, University of Twente, Enschede; BRUNO ANDREOTTI, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, ESPCI, Paris, DETLEF LOHSE, Physics of Fluids, University of Twente, Enschede — The existence and origin of line tension has remained controversial in literature. To address this issue we compute the shape of nanodrops using molecular dynamics and compare them using density functional theory in the approximation of the sharp kink interface. We show that the deviation from Young’s law is very small and would correspond to a typical line tension length scale (defined as line tension divided by surface tension) similar to the molecular size. It turns out that, for Lennard-Jones droplets, line tension is always negative and most pronounced at small contact angles. We propose an alternative interpretation based on the geometry of the interface at the molecular scale.

5:28PM EP.00007 Contact line dynamics of sessile nanofluid droplets under inert and saturated atmospheres. STUART JACK, KHELLIL SEFIANE, PRASHANT VALLURI, University of Edinburgh, OMAR MATAR, Imperial College London — We present experimental results concerning contact line dynamics of sessile ethanol droplets laden with TiO2 nanoparticles under unsaturated and saturated environments. The measuring apparatus comprises of a special motorised stage designed to allow for a range of forced speeds to study the dynamic effects. An isolated metallic chamber wherein the droplet was deposited allowed for maintenance of saturated or unsaturated conditions. Results show dependence of the driving force and the Capillary number (based on contact line velocity) - but, not in accordance with the traditional Cox-Voinov and hydrodynamic theories. Our analysis shows that the deviations from these standard theories could be due to local increase of viscosity at the contact line. Results also show that evaporation has little effect on the contact line behaviour of TiO2-Ethanol nanofluids.

5:41PM EP.00008 Optimization of Nanoparticle Separation through Solid State Nanopore. PRASHANTA DUTTA, TALUKDER JUBERY, Washington State University, ANMIV PRABHU, MINJUN KIM, Drexel University — Recently there has been a growing interest on solid state nanopores to separate biological molecules such as proteins, DNA, and RNA. However, efficient separation of biomolecules through nanopores is a challenging task as such factors as size and charge density of particle, size and charge density of membrane pore, and the concentration of bulk electrolyte influence the translocation behavior of nanoparticles through pores. To address this issue, a mathematical model is developed based on mass, momentum, and charge conservation equations to study the behavior of particles through pores. The surface charge density of the membrane pore was identified as the most critical parameter that determines the selectivity of the membrane and the throughput of the separation process. Based on this model, a single 150 nm pore was fabricated in a 50 nm thick free standing silicon nitride substrate by focused ion beam milling. This pore was subsequently chemically modified with (3-Aminopropyl) triethoxysilane to change its surface charge density. This chemically modified nanofluidic architecture was then used to separate 22 nm and 58 nm polystyrene nanoparticles.
5:54PM EP.00009 Separation of nanoparticles by flow past a patterned substrate, RUI ZHANG, JOEL KOPIKL, CCNY — We use molecular dynamics simulations to investigate trajectory deflection and particle trapping in flows of nanoparticle suspensions along patterned surfaces. Rigid atomicistic particles are suspended in a viscous liquid driven by a pressure gradient through a channel, one side of which has a pattern of alternating stripes which attract or repel the particles. The full wall interaction is obtained by summing over semi-infinite slabs of material with alternating van der Walls interactions, and has a non-trivial three-dimensional spatial variation. This wall interaction can either trap particles on the attractive stripes or deflect the trajectories of mobile particles away from the direction of mean flow. We determine how the motion of particles of different sizes is affected by the wall interactions, and in particular show that trajectory deflection is size dependent and that such flows may be used as “vector chromatography” separation technique.

6:07PM EP.00010 Spontaneous Imbition Dynamics of an n-Alkane in Nanopores: Evidence of Meniscus Freezing and Monolayer Sticking, PATRICK HUBER, SIMON GRUENER, Experimental Physics, Saarland University, D-66041 Saarbruecken (Germany), SAARLAND UNIVERSITY COLLABORATION — Capillary filling dynamics of liquid n-tetracosane (n-C24H50) in a network of cylindrical pores with 7 and 10 nm mean diameter in monolithic silica glass (Vycor) exhibit an abrupt temperature-slope change at $T_c = 54^\circ$C, $\sim 4^\circ$C above bulk and $\sim 16^\circ$C, $8^\circ$C resp., above pore freezing. It can be traced to a sudden inversion of the surface tension’s T-slope, and thus to a decrease in surface entropy at the advancing pore menisci, characteristic of the formation of a single solid monolayer of rectified molecules, known as surface freezing from macroscopic, quiescent tetracosane melts. The imbibition speeds, that are the squared prefactors of the observed square-root-of-time Lucas-Washburn invasion kinetics, indicate a conserved bulk fluidity and capillarity of the nanopore-confined liquid, if we assume a flat lying, sticky hydrocarbon backbone monolayer at the silica walls.

5:15PM EQ.00006 Digital Holographic Study of the Swimming Characteristics of Prorocentrum minimum (Dinophyceae)1, MYONG SOHN, KYUNG SEO, SANG LEE, POSTECH — The present study investigated the swimming characteristics of dinoflagellate Prorocentrum minimum, which is one of the cosmopolitan harmful algae species. A digital holographic PTV technique was employed to get the swimming trajectories of hundreds of P. minimum cells and to extract the kinematics of the flagella beating motion. The swimming speeds of P. minimum cells in a helical motion ranged from 20 to 140 µm · s⁻¹ and the average value of them was about 90±60 µm · s⁻¹. The mean value of the helix radius and pitch of the swimming trajectories were 3.8±1.6 µm and 34±15 µm, respectively. The longitudinal flagellum beating with a planar wave at the frequency of about 100 Hz. The transverse flagellum beating with a helical wave at the frequency of about 42 Hz. Effect of sea water viscosity was also analyzed. The increase of sea water viscosity reduced the flagella beating frequency and the swimming speed of P. minimum.

1This work was supported by Creative Research Initiatives (Diagnosis of Biofluid Flow Phenomena and Biomimic Research) of MEST/KRF.

5:28PM EQ.00007 Oscillatory Flows Induced by Swimming Microorganisms in Two-dimensions. JEFFREY S. GUASTO1, KARL A. JOHNSON2, J.P. GOLLUB3, Haverford College — We present the first time-resolved measurements of the oscillatory velocity field induced by swimming unicellular microorganisms. Confinement of the green alga Chlamydomonas reinhardtii in stabilized thin liquid films allows simultaneous tracking of cells and tracer particles. The phase-resolved velocity field reveals complex time-dependent flow structures, which evolve throughout the beat cycle of the organism, and the fluid velocity scales inversely with distance. The instantaneous mechanical power generated by the cells is measured from the velocity fields via the viscous dissipation and scales with the square of the swimmer speed. The peak power is about 15 fW, and the dissipation per cycle is more than four times what steady swimming would require. These observations carry important implications for the interpretation and modeling of transport processes, locomotion, and flagellar mechanics.

5:41PM EQ.00008 Swimming of a Microrobot Actuated by a Clinical Magnetic Resonance Imaging Apparatus. FREDERICK P. GOSSELIN, DAVID ZHOU, VIVIANE LALANDE, MANUEL VONTHRON, SYLVAIN MARTEL. Ecole Polytechnique Montreal — A miniature robot was designed to achieve fish-like locomotion when actuated by the imaging coils of a clinical Magnetic Resonance Imaging (MRI) system. The wireless fish robot is composed of a ferromagnetic head, a flexible tail and a float. The fish in an aquarium placed in the MRI, the robot is set into a swimming motion by an alternating transverse linear magnetic gradient. The influence of tail length, forcing frequency and forcing magnitude on the swimming velocity and flapping amplitude is investigated. Moreover, by using a combination of simultaneous magnetic gradients, the fish can reach superior swimming speeds than can be achieved by simply “pulling” on the fish with a magnetic field. Upon further miniaturization, the propulsion principle devised here could be used to navigate a micro surgical robot or a drug delivery system. A great advantage of this system is that no energy storage, motor or control system need to be carried by the robot, allowing great miniaturization possibilities.

5:54PM EQ.00009 Numerical simulation of a rotating elastic rod in a viscous fluid using the immersed boundary method. RAGHUNATH MANIYERI, YONG KWEON SUH, SANGMO KANG, Dong-A University, South Korea, MINJUN KIM, Drexel University, USA — Immersed boundary method has proved its efficacy in handling complex fluid structure interaction problems in the field of biological fluid dynamics. Inspired by the bacterial propulsion, we are interested to study the interaction of a rotating elastic cylindrical rod in a viscous fluid, where the flow is induced by the rotation of the rod. We developed a three dimensional computational model based on the immersed boundary method (IBM) in which Eulerian variables are used for the fluid flow and Lagrangian variables are used for the elastic rod motion. The Navier-Stokes equations governing the fluid flow are solved based on finite volume method on a staggered Cartesian grid system. The elastic rod is modeled by a number of circular rings with immersed boundary points on each ring. The motor part is modeled by a single circular ring at the base. We simulated for two cases- a straight and slightly bent rod and for an inclined rod. We found that for low rotational frequencies of the motor, the elastic rod undergoes simple axial rotation known as twisting motion and for high rotational frequencies it undergoes whirling motion with a discontinuous shape transition from straight to helical shape resulting into overwhirling. This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2010-0147).

6:07PM EQ.00010 Is paramecium swimming autonomic? . PROMODE R. BANDYOPADHYAY, NORMAN TOPOLSKY, JOSHUA HANSEN, Naval Undersea Warfare Center, Newport, RI 02841, USA — We seek to explore if the swimming of paramecium has an underlying autonomic mechanism. Such robotic elements may be useful in capturing the disturbance field in an environment in real time. Experimental evidence is emerging that motion control neurons of other animals may be present in paramecium as well. The limit cycle determined using analog simulation here could be used to navigate a micro surgical robot or a drug delivery system. A great advantage of this system is that no energy storage, motor or control system need to be carried by the robot, allowing great miniaturization possibilities. Sponsored by ONR 33.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session ER Drops IV Long Beach Convention Center 203C

4:10PM ER.00001 Tuning of the droplet motion in interconnected microfluidic devices . GUOQING HU, LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing, PR China, KUI SONG, LI ZHANG — The problem of controlling the droplet motions in multiphase flows on the microscale has gained increasing attention because the droplet-based microfluidic devices provide great potentials for chemical/biological applications such as drug discovery, chemical kinetics study, material synthesis, and DNA/cell assays. It is critical to understand the relevant physics on droplet hydrodynamics and thus control the generation, motion, splitting, and coalescence of droplets in complex microfluidic networks. The operation of those applications sometimes requires the arrival of droplets from different branch microchannels at a designated location within a transit time. We propose a simple design for interconnected microfluidic devices that implement the feedback mechanism to synchronize the droplet motion via a passive way. Numerical simulations using the Volume of Fluid (VOF) algorithm are conducted to investigate the time-dependent dynamics of droplets in both gas-liquid and liquid-liquid systems. An analytical mode based on the electronic-hydraulic analogy is also developed to describe the transit behavior of the droplet traffic. Both the numerical and experimental results agree well with the corresponding experimental results. Furthermore, we optimize the microfluidic networks to control the motion of a series of droplets.
or capsules to what is expected and observed for deformable drops. Capsules and vesicles show a decrease in the drainage time with the pushing force. Chesters [Chem. Eng. Res. Des. 69, 259-270 (1991)] that describes the time required to drain the thin, suspending fluid film that forms between two deformable droplets. Experiments were performed to demonstrate how changes in water pressure affect compound droplet water/oil volume ratio. In the method presented, oil is allowed to flow into the water nozzle with the pressure pulse ejecting both fluids as a compound droplet. Merged droplets. In an apparatus designed to produce single compound droplets, a piezoelectric diaphragm generates a pressure pulse from a voltage waveform input to eject a droplet. In the method presented, oil is allowed to flow into the water nozzle with the pressure pulse ejecting both fluids as a compound droplet. Experiments were performed to demonstrate how changes in water pressure affect compound droplet water/oil volume ratio.

**4:49PM ER.00004 Generation of Single, Monodisperse Compound Droplets**

JAMES BLACK, G. PAUL NEITZEL, Georgia Institute of Technology — Compound, nanoliter-scale droplets consisting of an aqueous inner phase surrounded by an oil encapsulant are of interest in a lab-on-a-chip process that levitates the droplets between a pair solid surfaces using therocapillary. The application requires a droplet with an oil-layer of sufficient thickness to permit the use of the levitation method, although not so thick as to impede effective combining and mixing of the contents of merged droplets. In an apparatus designed to produce single compound droplets, a piezoelectric diaphragm generates a pressure pulse from a voltage waveform input to eject a droplet. In the method presented, oil is allowed to flow into the water nozzle with the pressure pulse ejecting both fluids as a compound droplet. Experiments were performed to demonstrate how changes in water pressure affect compound droplet water/oil volume ratio.

---

**5:02PM ER.00005 ABSTRACT WITHDRAWN —**

---

**5:15PM ER.00006 A scaling theory for the hydrodynamic interaction between a pair of vesicles or capsules**

L. GARY LEAL, ARUN RAMACHANDRAN, UCSB Chemical Engineering — We present a scaling theory based on the analysis of A. K. Chesters [Chem. Eng. Res. Des. 69, 259-270 (1991)] that describes the time required to drain the thin, suspending fluid film that forms between two deformable capsules or vesicles as they are pushed towards each other by a constant force. Capsules and vesicles show a decrease in the drainage time with the pushing force, which results in the prediction that in a shear flow, the number of doublet formation events increases with the shear rate. Both trends are exactly opposite to what is expected and observed for deformable drops.

**5:28PM ER.00007 Sessile-drop oscillations fill a symmetry-breaking periodic table**

JOSHUA BOST-WICK, PAUL STEEN, Cornell University — Oscillations of a sessile drop are of fundamental interest for the contact-line instabilities they can exhibit and of practical importance in a number of industrial applications. We consider the small oscillations of the inviscid sessile drop under a number of contact-line conditions, including a contact-line modeled using a continuous contact-angle against speed relationship. The integro-differential equation, governing the motion of the interface, is formulated as a functional equation using inverse operators, which are parameterized by volume via the static contact angle of the drop base-state and by the mobility of the contact-line. In the symmetric limit, a hemispherical drop perturbed by a fixed contact-angle disturbance has characteristic oscillation frequencies, which are degenerate with respect to azimuthal wave-number much like the Bohr model of the atom is degenerate with respect to angular momentum quantum number. This degeneracy is broken by smoothly varying either i) the volume and/or ii) the contact line mobility. The analogy between the spectrum of these “broken” states and the filling order of the periodic table by energy levels both organizes and explains the hierarchy of frequencies.

**5:41PM ER.00008 Surface tension propulsion of fungal spores by use of microdroplets**

XAVIER NOBLIN, Laboratoire de Physique de la Matiere Condensee, CNRS - Universite de Nice Sophia Antipolis, Nice, France. SYLVIA YANG, Department of Biology, University of Washington, Seattle, USA, JACQUES DUMAIS, Department of Organismic and Evolutionary Biology, Harvard University, Cambridge, USA — Most basidiomycete fungi (such as edible mushrooms) actively eject their spores. The process begins with the condensation of a water drop at the base of the spore. The fusion of the droplet onto the spore creates a momentum that propels the spore forward. The use of surface tension for spore ejection offers a new paradigm to perform work at small length scales. However, this mechanism of force generation remains poorly understood. To elucidate how fungal spores make effective use of surface tension, we performed high-speed video imaging of spore ejection in Auricularia auricula and Saprobolomyces yeast, along with a detailed mechanical analysis of the spore ejection. We developed an explicit relation for the conversion of surface energy into kinetic energy during the coalescence process. The relation was validated with a simple artificial system.

**5:54PM ER.00009 Spreading Dynamics of a Droplet over a Superhydrophobic Surface**

NIKHIL BHOLE, CHARLES MALDARELLI — Aqueous droplet on a microtextured, superhydrophobic surface shows two distinct wetting behaviors, the Wenzel wetting and the Cassie-Baxter wetting. Most research efforts have focused on static energy arguments in which the overall surface energies of the Wenzel and Cassie-Baxter wetting states are compared to discern which is favored as a function of the surface topography and intrinsic surface energy. In this presentation we will construct a more relevant picture by examining the hydrodynamics of the wetting process on the scale of the topography. Our aim is to understand how the flow interacts with the topography to determine the wetting regime. We study the two dimensional spreading due to gravity of an aqueous drop over a well defined topographical pattern consisting of a periodic array of micron-sized posts. The flow in the droplet is assumed to be in the Stokes flow regime, and a boundary integral method is used for numerical solution with slip at the contact line and a velocity dependent relation for the dynamic wetting contact angle. The contact line shows a distinct slip-stick-jump (or slip-stick-penetration) motion over the topography determining the state of wetting.
essential for preventing catastrophic, irreversible wetting of superhydrophobic surfaces. We study the liquid droplet interaction on surfaces with different roughness configurations, ranging from a flat surface to a very rough surface. Very rough surfaces may induce the Cassie state, which is known to be a superhydrophobic state with a very low surface energy. Conversely, and depending on surface properties, surface texturing can also lead to a superhydrophilic state, which would be a beneficial condition for heat removal. The rough surfaces are heated and droplets are sprayed at high velocities using a droplet generator assembly. Visual inspections of droplet interactions with the different surfaces are carried out using a goniometer and a high speed camera.

Sunday, November 21, 2010 4:10PM - 6:20PM  
Session ES Drops V Long Beach Convention Center Grand Ballroom A

4:10PM ES.00001 Dynamics of water droplet on a superhydrophobic carbon nanotube array


4:23PM ES.00002 Spreading and breakup of a compound drop over a partially wetting substrate

Adrianus Aria, Christina Shu, Anirban Ghosh, Morzeza Gharib, Caltech — Among diverse types of superhydrophobic materials, arrays of vertically aligned carbon nanotube have attracted significant attention, mainly because of their exceptional properties. In this study, we look at the dynamic behavior of water droplet upon impact on the carbon nanotube array and subsequent primary rebound at wide range of Weber number. At small Weber number, the water droplet deforms and bounces off completely of the array, while at large Weber number, the water droplet splashes with fingering patterns. Our study shows that no pinning of the water droplet is observed on the array at both small and large Weber number, confirming that the water droplet does not have the capability to wet the array even at high impact velocity. In addition, the coefficient of restitution and spread factor of the bouncing and splashing water droplet will be discussed, along with the critical Weber numbers which predict when the droplet start to split and splash.

4:36PM ES.00003 Droplets on shallow grooved hydrophobic surfaces

Olesya Bliznyuk, Vasilisa Veligura, Stefan Kooij, Bene Poelesema, University of Twente — The equilibrium shape of water droplets on shallow-grooved hydrophobic surfaces is studied experimentally. The dependence of the two final states, notably metastable Cassie-Baxter and Wenzel, on the underlying geometric pattern is analyzed and discussed. The anisotropy of the patterns, including variation of the relative groove and ridge widths, allows studying the influence of different mechanisms of spreading in orthogonal directions and geometrical parameters on the final shape of the droplets. The validity of the Cassie-Baxter and Wenzel models in case of anisotropic surface is investigated comparing the experimental data with the theoretical predictions in the two respective regimes. The transition from one regime to another for different ridge width is discussed in relation to existing literature on this subject.

4:49PM ES.00004 Effects of the Secondary Length Scale on Hierarchical Superhydrophobic Surfaces Fabricated by Double-Layer Electron Beam Lithography

Jiansheng Feng, Jonathan Rothstein, University of Massachusetts — Surface topology is a key to superhydrophobicity. Many superhydrophobic surfaces found in nature have more than one characteristic roughness length scales. Very often the primary length scale is on the order of 10µm and the secondary length scale is on the order of 100nm. The secondary length scale is thought to play a key role in the stability and hysteresis of the hydrophobic surface. In our study, a novel method, double-layer electron beam lithography on SU-8 followed by surface sialisation on thermally deposited silicon dioxide coating, is used to fabricate superhydrophobic surfaces with micro-scale roughness and visible secondary length scale patterns. The feature size and spacing of the secondary patterns are varied to study their effects. The size and spacing of the primary scale features will also be varied independently, as well as the surface chemistry. Results of contact angle measurements and hysteresis will be presented.

5:02PM ES.00005 The Effect of contact angle hysteresis on droplet motion and collisions on superhydrophobic surfaces

Michael Nilsson, Jonathan Rothstein, University of Massachusetts Amherst — The effect of varying the contact angle hysteresis of a superhydrophobic surface on the characteristics and dynamics of water droplet motion and their subsequent collision are investigated using a high-speed camera. The surfaces are created by imparting random roughness to Teflon through sanding. With this technique, it is possible to create surfaces with similar advancing contact angles near 150 degrees, but with varying contact angle hysteresis. This talk will focus on a number of interesting experimental observations pertaining to drop collisions on surfaces with transition zones from one hysteresis to another, and the collision of droplets on surfaces of uniform hysteresis. For single drop studies, gravity is used as the driving force, while the collision studies use pressurized air to propel one drop into the other. For the case of droplet collision, the effect of hysteresis, Weber number, and impact number on the maximum deformation of the drops, and the post-collision dynamics will be discussed. For the single droplet measurements, the resistance to motion will be characterized as well as the transition from rolling to sliding as a function of drop size, inclination angle, and hysteresis. Additionally, we will quantify the effect of surface transitions on the resulting motion, mixing, and deflection of the drops.

5:15PM ES.00006 Wetting and Dewetting on Superhydrophobic Surfaces with Two-Tier Roughness

Jonathan Boreyko, Chuan-Hua Chen, Duke University, Mechanical Engineering and Materials Science Team — Many natural superhydrophobic structures, such as the lotus leaf, demonstrate hierarchical two-tier roughness. The hierarchical roughness is empirically known to promote robust superhydrophobicity, but the mechanism is still under debate. Here, we report the wetting and dewetting properties of two-tier roughness as a function of the wettability of the working fluid, where the surface tension of the water/ethanol mixture is tuned by the mixing ratio. On both natural and synthetic two-tier surfaces, externally deposited drops of increasing ethanol concentration exhibit two distinct wettin transitions, first for the impalement of the microscale texture and then for the nanoscale. The impaled drops are subsequently subjected to vibration-induced dewetting [1]. Drops impaling only the micro-scale roughness exhibit a bistable dewetting transition, similar to the Cassie state. In contrast, drops impaling both the micro and nano-scale roughness cannot be completely dewetted. Our work suggests that the nanoscale roughness is essential for preventing catastrophic, irreversible wetting of superhydrophobic surfaces.

5:28PM ES.00007 Dynamic wetting and hysteresis on superhydrophobic surfaces: an experimental observation of contact line motion. ADAM PAXSON, KATHERINE SMYTH, HYUK-MIN KWON, KRIPA VARANASI1. Massachusetts Institute of Technology — Contact angle and width are sampled at a high frequency to quantify advancing and receding behavior. As the contact angle increases, the contact line moves smoothly along the surface. As the contact angle recedes, instead of approaching a steady value, a stick-slip behavior occurs. The contact line sticks on the micro-pillars and forms capillary bridges, and slips when the bridges are stretched and then ruptured. The frequency of contact angle stick-slip behavior increased with contact line velocity. For the range of velocities tested, contact line velocity is not dependent upon pillar density, and does not appear to have an effect on measured contact angle values. This model of the moving contact line is verified by images captured using multiple methods. First, a silica nanoparticle solution is imaged under high magnification to observe contact line behavior during volume addition and subtraction. Additionally, the contact line of a sliding droplet is imaged with environmental scanning electron microscopy. This paper experimentally establishes for the first time advancing and receding behavior on micro-textured surfaces, and investigates the dependence of this behavior on contact line velocity.

5:41PM ES.00008 Droplet growth and coalescence on nanostructured surfaces during condensation. RYAN ENRIGHT, Massachusetts Institute of Technology, MATTHEW MCCARTHY, Drexel University, BENJAMIN HATTON, Harvard University, EVELYN WANG, Massachusetts Institute of Technology — In this work, we investigated the condensation behavior of water on nanostructured surfaces fabricated using a self-assembled virus template resulting in typical feature dimensions of 40 nm. These surfaces were first functionalized with a hydrophobic silane coating and, subsequently, some of the surfaces were selectively coated with hydrophilic PVA to create a chemically heterogeneous surface. The condensation process of water on these surfaces was characterized by microscopic imaging of the droplet growth behavior. The dynamics of energetic droplet coalescence events were obtained using high-speed imaging. Condensation on both the chemically homogenous and heterogeneous surfaces showed a preference for the unpinned Cassie droplet wetting mode. However, observed differences between the chemically homogenous and heterogeneous surfaces in both droplet growth and coalescence behavior demonstrate the effects of locally lowered nucleation energy barriers and increased droplet adhesion.

5:54PM ES.00009 Preferential Condensation of Water Droplets Using Hybrid Hydrophobic-Hydrophilic Surfaces. KRIPA VARANASI, MIT, TAO DENG, GE Global Research, ADAM PAXSON, RAJEEL D HUMAN, MIT — Heterogeneous vapor-to-liquid nucleation of water is an everyday phenomenon and plays an important role in the formation of rain drops, dew, heat transfer, water recovery, etc. Classical nucleation theory predicts that an energy barrier that depends strongly on the intrinsic wettability of the surface has to be overcome for the formation of initial liquid nuclei. Since the intrinsic wettability of regular surfaces is spatially uniform, heterogeneous nucleation of water droplets seems to occur in a random fashion without any particular spatial preference. This effect accounts for the recent observations on the loss of superhydrophobic properties of lotus leaves and associated synthetic surfaces under condensation. By taking advantage of the strong dependence of the nucleation energy barrier on wettability, we show for the first time that heterogeneous nucleation can be spatially controlled by the manipulation of the local intrinsic wettability of a surface. Using an environmental scanning electron microscope, we show that water droplets preferentially nucleate on the hydrophilic regions of the hybrid hydrophobic-hydrophilic surfaces we fabricated. Such ability to control water nucleation could address the condensation-related limitations of superhydrophobic surfaces and has implications for efficiency enhancements in energy, water, and electronics cooling systems.

6:07PM ES.00010 Deceleration-driven wetting transition of “gently” deposited drops on textured hydrophobic surfaces. KRIPA VARANASI, HYUKMIN KWON, ADAM PAXSON, MIT, NEELESH PATANKAR, Northwestern University — Many applications of rough superhydrophobic surfaces rely on the presence of droplets in a Cassie state on the substrates. A well established understanding is that if sessile droplets are smaller than a critical size, then the large Laplace pressure induces wetting transition from a Cassie to a Wenzel state, where liquid impales the roughness grooves. Thus, larger droplets are expected to remain in the Cassie state. In this work we report a surprising wetting transition where even a “gentle” deposition of droplets on rough substrates lead to the transition of larger droplets to the Wenzel state. A hitherto unknown mechanism based on rapid deceleration is identified. It is found that modest amount of energy, during the deposition process, is channeled through rapid deceleration into high water hammer pressure which induces wetting transition. A new “phase” diagram is reported which shows that both large and small droplets can transition to Wenzel states due to the deceleration and Laplace mechanisms, respectively. This novel insight reveals for the first time that the attainment of a Cassie state is more restrictive than previous criteria based on the Laplace pressure transition mechanism.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session ET Biolocomotion IV: Morphology of Flying and Swimming Long Beach Convention Center Grand Ballroom B

4:10PM ET.00001 Small, sleek, and in control: The body plan, sensory-neural control, and flight stability of insects. LEIF RISTROPH, Cornell University, ATTILA BERGOU, Brown University, JOHN GUCKENHEIMER, Z. JANE WANG, ITAI COHEN, Cornell University — Flying insects have evolved sophisticated sensory-neural systems, and here we argue that the fast reaction times of these systems reflect the need to overcome an intrinsic flight instability. We formulate a theory that shows how the body plan and flapping-wing aerodynamics determine the instability growth rate, which in turn dictates the response time needed to suppress it. We experimentally validate this theory by manipulating the flight, sensors, and body plan of fruit flies. The theory is general enough to describe a broad class of flying insects and also furnishes stability criteria for flapping-wing robots. Plausible body plans for the first flyers are determined by conjecturing that these insects were intrinsically stable and only later evolved fast-acting controls for the added benefit of flight agility.

4:23PM ET.00002 Drag Measurements over Embedded Cavities Modeled after Butterfly Scales in Low Reynolds Number Couette Flow. ROBERT JONES, AMY LANG, University of Alabama — Recent research has shown that symmetric, embedded square cavities can reduce the net drag acting on a surface through the formation of embedded vortices. It is hypothesized that the scales on butterfly wings (approximately 100 microns in length), though asymmetric, may act in a similar way resulting in greater flying efficiency. In this experimental study, cavities were modeled based on the geometry observed for bristled butterfly scales. Plates were designed to have parallelogram-shaped embedded cavities with an approximate 2:1 length to depth aspect ratio. The plates were suspended in high viscosity mineral oil above a rotating belt to generate a Couette flow condition such that the cavity Re was maintained in a similar regime as that occurring for the flow over butterfly scales. The net drag forces were measured with a force gauge and compared to flat plate measurements in the same facility. The variation in drag over a range of Reynolds numbers was analyzed.

1Corresponding Author

Funding from a University of Alabama RGC grant is gratefully acknowledged.
4:36PM ET.00003 The Effect of Wing Scales on Monarch Butterfly Flight Characteristics1. ANGELA SHAW, Northridge High School, ROBERT JONES, AMY LANG, University of Alabama — Recent research has shown that the highly flexible wings of butterflies in flapping flight develop vortices along their leading and trailing edges. Butterfly scales (approximately 100 microns in length) have a shingled pattern and extend into the boundary layer. These scales, which make up approximately 3% of the body weight or less, could play a part in controlling separation and vortex formation in this unsteady, three-dimensional complex flow field. A better understanding of this mechanism may lead to bio-inspired applications for flapping wing micro-air vehicles. In this study, the flight performance of Monarch (Danaus plexippus) butterflies with and without scales was analyzed. Scales were removed from the upper and lower wing surfaces and specimens were videotaped at 600 frames per second. Variation in flapping patterns and flight fitness were observed.

1Work performed as a Research Experience for Teachers REU site sponsored by NSF grant EEC 1035239.

4:49PM ET.00004 Explanation of the effects of leading-edge tubercles on the aerodynamics of airfoils and finite wings1. MEHDI SAADAT, HOSSEIN HAJ-HARIRI, University of Virginia, FRANK FISH, West Chester University — A computational study was conducted to explain the aerodynamic effect of leading edge tubercles on maximum lift coefficient, stall angle of attack, drag, and post stall characteristics for airfoils as well as finite wings. Past experiments demonstrated airfoils with leading edge tubercles do not improve $C_{\text{max}}$, drag, or stall AoA but smoothen post stall characteristics to a great degree. In contrast to airfoils, finite wings with L.E. tubercles improved all aerodynamic characteristics. We explain the stall mechanism of the tubercled wing by considering each L.E. tubercle as a combination of a swept forward and a swept backward wing. There are 3 mechanisms (streamline curvature, accelerated stall, and upwash) that cause $C_{\text{max}}$ of airfoils with L.E. tubercles always be lower than that of smooth airfoils. We also identify two additional mechanisms which are responsible for improved post-stall characteristics of airfoils with L.E. tubercles. Finally, we discuss why finite wings with L.E. tubercles have higher $C_{\text{max}}$ and lower drag than their smooth L.E. counterparts by studying effects of wing tip, sweep, and taper ratio.

5:02PM ET.00005 Hydrodynamics of penguin wing models. FLAVIO NOCA, hepia, hepia, NHUT CUONG DUONG, EPFL, JEROME HERPICH, hepia — The three-dimensional kinematics of penguin wings were obtained from movie footage in aquariums. A 1:1 scale model of the penguin wing (with an identical planform but with a flat section profile and a rigid configuration) was actuated with a robotic arm in a water channel. The experiments were performed at a chord Reynolds number of about $10^4$ (an order of magnitude lower than for the observed penguin). The dynamics of the wing were analyzed with force and flowfield measurements. The two main results are: 1. a net thrust on both the upstream and downstream movement; 2. the occurrence of a leading edge vortex (LEV) along the wing span. The effects of section profile, wing flexibility, and a higher Reynolds number will be investigated in the future.

5:15PM ET.00006 The Performance of Finite-span Hydrofoils with Humpback Whale-like Leading Edge Protuberances1. DERRICK CUSTODIO, Worcester Polytechnic Institute, CHARLES HENOC, Naval Undersea Warfare Center, Newport, RI, HAMID JOHARI, California State University, Northridge — The effects of leading edge protuberances on the lift and drag performance of finite-span hydrofoils were examined in a series of water tunnel experiments. The leading edge protuberances are analogous to the tubercles on humpback whale pectoral flippers. The hydrofoils have a rectangular planform and an aspect ratio of 4. The hydrofoil section profile is based on NACA 63(4)-021, and the leading edge has a sinusoidal geometry with constant amplitude and wavelength. The hydrofoil angle of attack was varied up to 30°, and the freestream velocity ranged from 1.8 to 5.4 m/s. Results indicate that the hydrofoils with leading edge protuberances do not stall in the traditional manner. Below 12° lift increased linearly with angle of attack. Beyond this angle, the lift either attained a nearly constant value or increased slowly up to 30° depending on the Reynolds number. Drag increased continuously with the angle of attack, and was not dependent on the Reynolds number. These observations are consistent with our previous infinite span hydrofoil data, and may be explained in terms of the flow modifications created by the leading edge protuberances.

1Supported through MURI ONR N00014-08-1-0642.

5:28PM ET.00007 Why fishes have a fish shape. CHRISTOPHE ELOY, LIONEL SCHOUVEILIER, IRPHE, Marseille, France — The relation between form and function for elongated swimmers is revisited by solving a multi-objective optimization problem. We consider elongated fishes of varying elliptic cross-section whose motion is prescribed by a time-periodic curvature. The two semi-axes of the cross-section, the curvature amplitude and phase are assumed to vary continuously along the fish length. Hydrodynamic forces acting on such fishes are modeled in the elongated-body limit by considering both reactive and resistive forces. Applying Newton’s second law, the heave and pitch amplitude and phase, as well as the swimming velocity can be found. The total power needed can also be calculated yielding the swimming efficiency. The multi-objective optimization consists in finding the fish shape and associated motion which corresponds to maximum efficiency, maximum velocity or any trade-off between the two. This optimization problem is solved using a genetic algorithm whose principle is to start with an initial random population and to evolve it by mutation and selection. We find that the most efficient shape resembles existing fishes and arguments are given to explain the relation between this particular fish form and performance.

5:41PM ET.00008 Viscous to inertial pumping transitions in a robotic gill plate array1. MARY LARSON, KEN KIGER, University of Maryland — Biological oscillating appendage systems are known to exhibit distinct patterns of movement based on their Reynolds number. Flapping kinematics (net flow perpendicular to appendage stroke plane) are associated with Re > 100, while rowing kinematics (flow in the direction of appendage motion) are typically associated with Re < 1. Previous studies of pumping by mayfly gill plate arrays have shown a transition between rowing and flapping at a Re $\approx$ 5. Although the flow generated by the animal could be documented, the limited range of behavior of the animal prevented a detailed study of why and how such a pumping mechanism might be optimized. Towards this end, a two-degree-of-freedom robotic oscillating plate array has been constructed, which allows for the variation of the Reynolds number, plate spacing, plate shape, and stroke/pitch amplitude beyond what is exhibited by the animal system. Using PIV, these combinations allow the individual influence of each feature on the pumping efficiency to be observed, and elucidate how it may be optimized for engineered devices. The current results will compare this simplified system to the flow generated by the typical mayfly, to determine how effectively the model performs in comparison to the more complex animal system.

1The authors acknowledge support from the NSF grant CBET-0730907.
the simulation data. In addition to shear layer instability we analyze Rayleigh-Taylor as a potential instability mechanism of the liquid column.

wavelength of the shear layer instability following the procedure of Boeck & Zaleski (2005). The theoretical wavelengths are comparable to those extracted from averaging of the simulation data we extract mean interface geometries and boundary layer velocity profiles. These are used to calculate the most unstable liquid column, we perform proper orthogonal decomposition of side view images extracted from detailed simulations of the near injector primary atomization and augmentors. The mechanisms by which the liquid jet initially breaks up, however, are not well understood. To analyze the instability mechanism of the liquid and gaseous phases, supplemented by singular source terms that properly account for thermal expansion effects. The numerical scheme has been tested of two–phase flow is computationally studied within the context of a hydrodynamic theory. A level-set method is used to track down the phase interface, which

This paper is intended to demonstrate the capability of a numerical approach to investigate the primary atomization process. The evaporation on the interface, collapse to-collision distance, and collapse phase lags on the net flow rate, pressure gradient, wall shear stress, velocity are investigated.

Sunday, November 21, 2010 4:10PM - 6:20PM — Session EU Multiphase Flows III Hyatt Regency Long Beach Regency A

4:10PM EU.00001 Predicting Biomass Fluidization through Appropriate Modeling of Initial Conditions, FRANCINE BATTAGLIA, JONAS ENGLAND, SANTHIP KANHOHY, MIRKA DEZA, Virginia Polytechnic Institute and State University — Fluidized bed gasifiers can be used to convert feedstock with low-carbon content into fuels, basic chemicals and hydrogen. When biomass is the feedstock, there is a difference in the fluidization behavior between the solid particles and bed media (e.g., refractory sand) due to contrasting size, shape and particle density. The differences can lead to poor solid fuel distribution and diminished gasifier performance. The present work will focus on computational simulations of a fluidized bed gasifier using an Eulerian-Eulerian model to represent the gas and solid phases as interpenetrating continua. Recent studies to predict biomass fluidization motivated this study to reassess how to best model gas-solid characteristics that capture the same physics measured experimentally. Relations for pressure drop are used to correct for the bed mass by either adjusting the initial solids packing fraction or initial bed height, two parameters that must be specified in CFD models. It was found that adjusting the initial solids volume packing correctly predicted the pressure drop measured experimentally but underpredicted the minimum fluidization velocity. By adjusting the initial bed height to correct for the mass, both the pressure drop and minimum fluidization velocity were successfully predicted without artificially altering the physics and retaining the known characteristics of the bed material.

4:23PM EU.00002 Numerical study of impact of evaporation on liquid jet in cross-flow, MARIOS SOTERIOU, XIAOYI LI, MARCO ARIENTI, United Technologies Research Center — Atomization of a liquid fuel jet by a high speed cross-flowing gas plays a critical role in many propulsion devices. High fidelity simulation offers the potential of a better understanding and enhancement of this atomization process. In this work, a computationally efficient hybrid Eulerian-Lagrangian approach is coupled with a droplet evaporation model and is used to probe the impact of evaporation on the spray development. The Coupled Level Set and Volume of Fluid (CLSVOF) method is used to directly calculate the breakup and coalescence of the liquid-gas interface. Adaptive Refinement (AMR) is adopted to achieve high resolution at the interface. Small fuel droplets in dilute regions are removed from the Eulerian description, transformed into Lagrangian particles and tracked by a discrete phase transport model. The coupling of the spray evaporation to the gas phase is examined with respect to jet blockage, spray penetration, and overall far-field spray dispersion. The calculation is validated with flow rate, spray size distribution and velocity data acquired in a spray rig at high-Weber, high-Reynolds number injection conditions. The effect of evaporation on spray distribution is also discussed.

4:36PM EU.00003 Characterization of primary atomization mechanism of straight liquid jets, JUNJI SHINJO, Japan Aerospace Exploration Agency, AKIRA UMEMURA, Nagoya University — Detailed numerical simulations of straight liquid jets have been carried out to elucidate the mechanism of liquid primary atomization. The impact of liquid against the gas forms the umbrella-shaped front where initial atomization occurs subsequently. At later times, surface instability also develops on the liquid core surface, leading to ligament/droplet formation from the core. As the ligament/droplet formation mechanism has been already reported, this study mainly focuses on this surface instability development. Disturbances for instability development come from the front through the recirculating gas flow and from the gas-liquid interaction of the core itself. Several cases are compared to identify the parameter dependence of instability development and the results are compared with the theoretical prediction.

4:49PM EU.00004 Modeling and simulation of primary atomization with phase transition, PENG ZENG, BERND BINNINGER, NORDERT PETERS, HEINZ PITSCH, RWTH Aachen University, MARCUS HERRMANN, Arizona State University — Atomizing liquids by injecting them into crossflows is a common approach to generate fuel sprays in gas turbines and augmentors. The mechanisms by which the liquid jet initially breaks up, however, are not well understood. To analyze the instability mechanism of the liquid column, we perform proper orthogonal decomposition of side view images extracted from detailed simulations of the near injector primary atomization region. This analysis shows a dominant wavelength with the associated interface corrugation traveling downstream with the jet. Using consistent temporal averaging of the simulation data we extract mean interface geometries and boundary layer velocity profiles. These are used to calculate the most unstable wavelength of the shear layer instability following the procedure of Boeck & Zaleski (2005). The theoretical wavelengths are comparable to those extracted from the simulation data. In addition to shear layer instability we analyze Rayleigh–Taylor as a potential instability mechanism of the liquid column.

5:02PM EU.00005 Analysis of Column Instability Modes in Liquid Jet in Crossflow Atomization, SINA GHODS, Arizona State University, MARCO ARIENTI, MARIOS SOTERIOU, United Technologies Research Center, MARCUS HERRMANN, Arizona State University — Atomizing liquids by injecting them into crossflows is a common approach to generate fuel sprays in gas turbines and augmentors. The mechanisms by which the liquid jet initially breaks up, however, are not well understood. To analyze the instability mechanism of the liquid column, we perform proper orthogonal decomposition of side view images extracted from detailed simulations of the near injector primary atomization region. This analysis shows a dominant wavelength with the associated interface corrugation traveling downstream with the jet. Using consistent temporal averaging of the simulation data we extract mean interface geometries and boundary layer velocity profiles. These are used to calculate the most unstable wavelength of the shear layer instability following the procedure of Boeck & Zaleski (2005). The theoretical wavelengths are comparable to those extracted from the simulation data. In addition to shear layer instability we analyze Rayleigh–Taylor as a potential instability mechanism of the liquid column.

1 This work was supported by NSF grant CBET-0853627 and NavAir SBIR N07-046.
of the Taylor-lengthscale size, i.e. the values of their volume fraction, and mass fraction. Here, \( \tau \)

\( \text{GHOBASHI, University of California, Irvine} \) — It has been established both experimentally and numerically (e.g. Ferrante and Elghobashi (Phys. Fluids 2003)), that for different test cases have identical Stokes number and volume fraction, they have different effects on the turbulence kinetic energy, \( \tau \)

\( \text{F. LUCCI, University of California, Irvine, A. FERRANTE, University of Washington, S. EL-} \)

DNS. Two cases with different air-flow rates are performed to investigate the mechanism and stability of air layer. For high air-flow rate, the stable air layer is

\( \text{DNS. The level set method is used to track the phase interface and the structured-mesh finite volume solver is used with an efficient algorithm for two-phase} \)

\( \text{section length before the step is 3h and the post expansion length is 30h, where h} \)

\( \text{the total number of grid points is about 271 million for} \)

\( \text{into turbulent water flow for ALDR. The Reynolds and Weber numbers based on the water properties and step height are 22,800 and 560, respectively. An inlet} \)

\( \text{observed a stable air layer on an entire flat plate if air is injected beyond the critical air-flow rate. In the present study, air is injected at the step on the wall} \)

\( \text{and driven sessile drops. We investigate the critical conditions for the transition among the regimes as affected by substrate wettability, initial} \)

\( \text{— Results will be presented for pressure-driven turbulent gas flow over a liquid layer in a 3D channel. These have been obtained with a DNS code that resolves} \)

\( \text{any connectivity. This approach combined with the original front tracking concept allow us to model bubble topological changes automatically. By letting the} \)

\( \text{such connectivity reconstruction can be quite expensive. In this work, we adopt the point-set method\cite{1} to construct each individual interfacial point without} \)

\( \text{point-set front tracking method with finite element, CHU WANG, LUCY ZHANG — The capability of handling constant and} \)

\( \text{to be at a constant level, the indicator field is smeared out using the quintic B-Spline function. A regeneration method adopting one-dimensional} \)

\( \text{in order to cope with the topology change. The interface points are then coupled with a finite element fluid} \)

\( \text{— Results will be presented for pressure-driven turbulent gas flow over a backward-facing step}\cite{1}. DOKYUN KIM, PARVIZ MOIN, Stanford University — Direct Numerical Simulation (DNS) of two-phase flow is performed to investigate} \)

\( \text{all discontinuities across the gas/liquid interface in a sharp manner. In the simulations considered here, the interface is forced to remain flat; the corresponding} \)

\( \text{wall-type region near the interface is therefore resolved. Mean profiles near the interface obtained with this method will be used to assess to what extent the turbulence can be represented by near-wall turbulence. The results for} \)

\( \text{for the distribution of shear stress exerted by the gas on the liquid layer have implications on large-scale modeling of turbulent two-phase flows.} \)

\( \text{Interfacial flows in micro-channels: flow regimes and transitions. MAJID AH-} \)

\( \text{MAJID AH-MADLOUYDARAB, PENG GAO, JAMES J. FENG, University of British Columbia — We report simulations of gas-liquid flows in periodically patterned} \)

\( \text{micro-channels with grooves and ridges. A diffuse-interface model is used to handle the interfacial motion and the three-phase contact line. A constant body} \)

\( \text{depending on the competition between the driving force and capillary force and the level of} \)

\( \text{in the micro-channel, including slug flows with air bubbles, slug flows with water drops, water rivulets alongside air flow and driven sessile drops. We investigate the critical conditions for the transition among the regimes as affected by substrate wettability, initial} \)

\( \text{morphology of the interface, geometry of the micro-channel and viscosity ratio.} \)

\( \text{Sunday, November 21, 2010 4:10PM - 6:20PM} \)

\( \text{Session EV Particle Laden Flows: Simulations I} \)

\( \text{ABSTRACT WITHDRAWN} \)

\( \text{ABSTRACT WITHDRAWN} \)

\( \text{ABSTRACT WITHDRAWN} \)

\( \text{Is Stokes number an appropriate indicator for turbulence modulation by particles of Taylor-length-scale size?} \)

\( \text{F. LUCCI, University of California, Irvine, A. FERRANTE, University of Washington, S. EL-} \)

\( \text{GHOBASHI, University of California, Irvine — It has been established both experimentally and numerically (e.g. Ferrante and Elghobashi (Phys. Fluids 2003)), that the Stokes number,} \)

\( \text{can be used as an indicator to determine the extent to which small particles,} \)

\( \text{modify the turbulence structure, for fixed values of their volume fraction, and mass fraction. Here,} \)

\( \text{We employ DNS with an immersed boundary method to fully resolve the flow around thousands of freely} \)

\( \text{of Taylor-lengthscale size \((d_p \sim \lambda)\) in decaying isotropic turbulence with initial} \)

\( \text{Our results show that although the particles in different test cases have identical Stokes number and volume fraction, they have different effects on the turbulent kinetic energy,} \)

\( \text{resulting in a faster decay of} \)

\( \text{is not an appropriate indicator for determining the extent of turbulence modulation by particles with} \)

\( \text{Supported by the Office of Naval Research} \)

\( \text{— EV.00007 Simulations of bubble coalescence and breaking-up using connectivity-free} \)

\( \text{— EV.00008 Direct Numerical Simulation of Air Layer Drag Reduction over a Backward-facing} \)

\( \text{DOKYUN KIM, PARVIZ MOIN, Stanford University — Direct Numerical Simulation (DNS) of two-phase flow is performed to investigate} \)

\( \text{air layer drag reduction (ALDR) phenomenon in turbulent flow over a backward-facing step. In their experimental study, Elbing et al. (JFM, 2008) have observed} \)

\( \text{DNS. The level set method is used to track the phase interface and the structured-mesh finite volume solver is used with an efficient algorithm for two-phase} \)

\( \text{two-phase DNS. Two cases with different air-flow rates are performed to investigate the mechanism and stability of air layer. For high air-flow rate, the stable air layer is} \)

\( \text{air is formed on the plate and more than 90% drag reduction is obtained. In the case of low air-flow rate, the air layer breaks up and ALDR is not achieved. The} \)

\( \text{parameters governing the stability of air layer from the numerical simulations is also consistent with the results of stability analysis.} \)
Meshkov instability prior to, and following, reshock, JEFFREY JACOBS, OLEG LIKHATCHEV, VLADIMER TSIKLASHVILI, rate occurs after a very short period of time. is to determine the dependence of the post reshock growth rate on the reflected shock strength and whether the growth rate is independent of the mixing zone microscopically.

Gradient waves are generated through the excitation of Faraday internal waves using two synchronized loudspeakers that oscillate the gases vertically. The study focuses on the University of Arizona — Richtmyer-Meshkov instability is investigated utilizing shock tube experiments. The instability is generated by the impulsive acceleration of dense solid particles such as gas-fluidized beds, particles take complex arrangements as a result of interactions with surrounding particles, walls and gas flows. This kind of structure formation gives large influence on the overall flow behavior. Due to the existence of dense particles, it is still difficult to investigate the microscopic flows occurring in the narrow gaps in-between particles accurately and it has not been discussed well up to the present. In the present study, a direct numerical simulation by coupling discrete element method (DEM) and immersed boundary method (IBM) is performed. This is the first step of our continuing study and the drag force working on particles and permeability of flows are investigated in fixed bed conditions. In addition to the particle arrangement, the volume fraction and Reynolds number are varied and its influence is discussed. The results of a two-dimensional gas-fluidized bed are also presented in the study.

Particle Dispersal in Rapid Expanding Gas Flow: Importance of Unsteady Force and Heat Transfer, YUE LING, ANDREAS HASELBACHER, S. BALACHANDAR, University of Florida — When a highly compressed gas-particle mixture is suddenly released, the particles will be dispersed outward in very high speed driven by the rapid expanding gas. This is a phenomenon which can be observed in nature, such as volcanic eruption, and many industrial applications, such as detonation of multiphase explosives. The unsteady compressible nature of the gas flow coupled with the motion of dispersing particles makes accurate prediction of particle behavior challenging. The unsteady force and heat transfer becomes very important in the momentum and energy transfer between the gas and particle phases. A multiphase flow model is suggested to simulate this problem by Eulerian-Lagrangian approach. The significance of the unsteady force and heat transfer to the quasi-steady force and heat transfer are first justified by using the presented model to solve the problem of shock-particle interaction. The peak values and the total impulses of different forces and heat transfer shows that the unsteady force and heat transfer are important. Then the multiphase model will be used to solve the problem of particle dispersion in rapid expanding flow. This problem can be viewed as an extension of the classic problem considered by Brode H. L. (J. Appl. Phys., Vol. 26, 1955, pp. 766–775). One-way coupled simulations are carried out for several initial conditions and particle properties.

Extension of Basset-Boussinesq-Oseen and Maxey-Riley Equations to Compressible Flows, MANOJ PARMAR, ANDREAS HASELBACHER, S. BALACHANDAR, University of Florida — Viscous compressible flow around a sphere is considered in the limit of vanishing Reynolds and Mach numbers. Using the analytical solution derived in earlier works, an exact expression for the transient force on a sphere in uniform flows is presented. The transient force is decomposed into quasi-steady, inviscid unsteady, and viscous unsteady components. The influence of compressibility on each of these components is examined. Numerical results for the transient force are in excellent agreement with theory. The present formulation thus offers an explicit expression for the unsteady force in the time domain and can be considered as a generalization of the Basset-Boussinesq-Oseen equation to the compressible flow regime that can be used in numerical simulations of compressible multiphase flows. An extension of Maxey-Riley equation for particle motion in non-homogeneous compressible flows is also proposed.

Tracking Rigid Spherical Particles in Incompressible Flows via Dissipative Hydrodynamics, BRENT HOUCHENS, KENNETH DAVIS, Rice University, YONG SHI, ALBERT KIM, University of Hawaii at Manoa — Solutions for particle trajectories computed using dissipative hydrodynamics (DHD) for rigid spherical particles are discussed. DHD reproduces the many-body hydrodynamics of Stokesian Dynamics (SD), but is more computationally efficient. DHD satisfies the fluctuation-dissipation theorem of Dissipative Particle Dynamics (DPD) and therefore is not hindered by the relaxation-time limitations of Stokesian Dynamics. For a given continuum flow field, the translations and rotations of multiple particles are calculated taking into account both stochastic dissipative effects and deterministic conservative forces. Examples of particle tracking in two-dimensional and three-dimensional flows, computed via spectral element simulations, will be discussed.

LBM Simulations of 3D Peristaltic Transport with Particles, KEVIN CONNINGTON, The Benjamin Levich Institute, The City College of New York, QINJUN KANG, HARI VISWANATHAN, Los Alamos National Labs, EES-16 Division, SHIYI CHEN, The Johns Hopkins University, Department of Mechanical Engineering — “It is sometimes necessary to produce a flow through a duct without using internal moving parts such as rotors or pistons. This need may arise when the fluid is corrosive or toxic, or when the fluid carries solid particles for which a passage free of obstacles would be desirable.” (Hanin, 1968) Peristaltic transport offers a suitable solution by eschewing the use of internal flow drivers. A peristaltic flow occurs when a tube with flexible walls transports the contained material by progressing a series of contraction waves along the length of those walls. The deformation induces pressure gradients which drive the flow. Although significant progress has been made to provide the theory and analysis of peristaltic flows, relatively little research has been performed on the effects of particle transport due to the complexities involved. The Lattice Boltzmann Method provides a convenient formulation of peristaltic flows through numerical simulation. This talk investigates the transport and behavior of particles in a model peristaltic system by varying the relevant dimensionless parameters of the problem. It is found that the peristaltic transport is maximized for situations where the peculiar phenomenon of “trapping” is realized.

Resolved particle direct numerical simulation of dense particle-laden flows is used to investigate gas-particle interactions in the case of moving and colliding particles. Numerics are based on an immersed boundary implementation that discretely conserves mass and momentum (Meyer et al., JCP 2010). Geometry is accounted for through fictitious cut cell algorithm. Fluxes are rescaled based on gas face and gas cell fractions, while second order accurate volume and surface calculations are obtained through marching tetrahedra reconstruction of the interface. This approach relaxes flow blockage due to the immersed boundaries and is consistent with the soft sphere model used for particle collisions. A Lagrangian solver describes the particle motion. Conservation of momentum across phases is ensured since the forces applied to the particles are directly derived from the Navier-Stokes source terms. This new tool allows investigating the accuracy of existing drag models for freely moving particles (finite Stokes) and colliding particles (sudden and frequent direction changes).
4:36PM EW.00003 Turbulence in reshocked Richtmyer-Meshkov unstable fluid layers, B.J. Balakumar, Greg Orlicz, Ray Ristorcelli, Sridhar Balasubramanian, Kathy Prestridge, Chris Tomkins, Los Alamos National Laboratory — Advances in the implementation of high resolution PIV (150um vector-to-vector resolution) and PLIF (50um resolution) diagnostics have allowed the experimental measurement and characterization of turbulent mixing in Richtmyer-Meshkov unstable fluid layers after reshock (Balakumar et. al., Phys. Fluids, 2008). Using instantaneous PLIF data obtained at closely spaced intervals of time, we illustrate the rapid disintegration of the primary wavelengths of the initial interface and the beginning of a turbulence cascade generating smaller flow structures after reshock. The enhanced mixing is reflected in the variation of the density probability distribution function between the pre-reshock and post-reshock states. The density self-correlation is observed to exhibit a double-peaked structure and mild non-Boussinesq effects are observed in a layer with varicose initial interfacial perturbations. Density and velocity pdfs are used to examine the streamwise asymmetry of the mixing layer with large fluctuations occurring preferentially upstream of the centerline. Other turbulence statistics including the 2nd and 4th order structure functions, RMS statistics (both velocity and density) and turbulence intensity are also presented.

4:49PM EW.00004 Simulation of Material Mixing in Shocked and Reshocked Gas-Curtain Experiments1, Akshay Gowardhan, Fernando Grinstein, LANL — The unique combination of shock and turbulence emulation capabilities supports direct use of implicit large eddy simulation (ILES) as an effective simulation anzatz in shock-driven mixing research. This possibility is demonstrated in the context of a prototypical case study for which available laboratory data can be used to test and validate the ILES modeling. The particular ILES strategy tested here is based on a nominally-inviscid simulation model using LANL’s RAGE code and adaptive mesh refinement. An SF6 gas curtain is formed by forcing SF6 through a linear arrangement of round nozzles into the shocktube test section. Once a steady state is achieved, the gas curtain is shocked (M=1.2), and its later evolution subject to Richtmyer-Meshkov flow instabilities, transition, and non-equilibrium turbulence phenomena are investigated based on high resolution simulations for shocked and reshocked cases. The gas curtain used at initialization for the RAGE simulations was separately simulated using a 3D Navier-Stokes-Boussinesq code. Various approaches to introducing weak 3D perturbations to emulate the noise inherent in the experimental setup were tested with special focus on addressing initial condition effects on late-time mixing.

1Funded by the LANL LDRD-ER on “Turbulence by Design.”

5:02PM EW.00005 Modeling the Richtmyer-Meshkov Instability through Baroclinic Vorticity Production, Christopher Weber, Riccardo Bonazza, University of Wisconsin-Madison, Andrew Cook, Lawrence Livermore National Laboratory — The Richtmyer-Meshkov instability (RMI) is modeled using the baroclinic term in the vorticity transport equation and the results are compared to numerical simulations. The baroclinic vorticity production equation, \( \omega_t = (\nabla \times \rho \nabla p) / \rho^2 \), is simplified using an impulsive hydrostatic pressure gradient. Using this approximation, one can calculate the vorticity field from an initial density field and the post-shock 1D velocity. An analytic equation for circulation and perturbation growth rate is found for a single mode interface which proves accurate up to moderate Atwood numbers and Mach numbers. This model also accurately predicts the behavior of a more complicated problem with an arbitrary density field. The growth rate from the compressible RMI simulation and an incompressible simulation initialized with the same initial density field and the model’s velocity field compare very favorably to low Mach numbers.

5:15PM EW.00006 Sensitivity of Shock Accelerated Multi-Component Compressible Flows1, Santosh Shankar, Sanjiva Lele, Stanford University — Numerical simulation of Richtmyer-Meshkov instability (RMI) is conducted using an improved localized artificial diffusivity (LAD) method which is used to treat discontinuities in the form of material-interfaces and shocks in the flow-field. The RMI occurs on a cylindrical interface between air and SF6 accelerated by a Mach 1.2 shock initially in air. Navier-Stokes simulation is conducted to accurately predict the mixing between the two fluids. The initial conditions for the 2-D simulations are matched to previous experimental work by Tomkins et al (JFM, 2008). Sensitivity of the mixing rate to mesh resolution is found to be higher at grid converged results. The study on initial condition sensitivity indicates that the initial pressure and density gradient are critical parameters which determine the primary vortex generation responsible for the flow development. The effect of presence of the third species (acetone used as a tracer particle in the experiments to obtain contour fields using PLIF) is shown to be non-negligible and an estimate of the amount of the tracer species that was present in the initial experimental set-up is given.

1Work supported by DOE-SciDAC Grant DE-FC02-06-ER25787.

5:28PM EW.00007 Effects of Initial Conditions on the Planar Richtmyer-Meshkov Instability1, Fernando Grinstein, Akshay Gowardhan, LANL — In the large eddy simulation (LES) approach, large-scale energy-containing structures are resolved, smaller structures are filtered out, and unresolved subgrid effects are modeled. Extensive recent work has demonstrated that predictive under-resolved simulations of the velocity fields in turbulent flows are possible without resorting to explicit subgrid models, when using a class of physics-capturing high-resolution finite-volume numerical algorithms. This strategy is denoted implicit LES (ILES) [1]. Tests in fundamental applications ranging from canonical to complex flows indicate that ILES is competitive with conventional LES in the LES realm proper—flows driven by large scale features. The performance of ILES in the substantially more difficult problem of under-resolved material mixing driven by under-resolved velocity fields and initial conditions is the focus of the present work. Progress in addressing relevant resolution issues in RAGE simulations of planar shocked and reshocked driven turbulence is reported.

1Funded by the LANL LDRD-ER on “Turbulence by Design.”

4:23PM EW.00002 Nonideal Effects in Single Mode Richtmyer-Meshkov Instability Shock Tube Experiments, Robert Morgan, Jeffrey Jacobs, The University of Arizona, Jeffrey Greenough, William Cabot, Lawrence Livermore National Laboratory — Shock tube experiments on the late time Richtmyer-Meshkov instability (RMI) are presented. The growth of the instability from a diffuse sinusoidal initial perturbation impacted by a Mach 1.2 shock wave (SW) is studied. The RMI develops from an air-SF6 interface in a 2.0 m long test section. The RMI is visualized using planar Mie scattering using an Nd:YLF laser for illumination and recorded using high speed CMOS cameras. This visualization system allows the recording of the time history of the RMI. Measured growth rates are found to be greater than those predicted by models and numerical simulations. In addition, measurements of SW accelerated flat interfaces show them to accelerate after SW interaction. A numerical investigation was then undertaken to investigate the effects of boundary layers and openings in the shock tube test section. These simulations show that openings tend to reduce the impulsive drive affecting the interface but still produce constant interface velocity. However, the presence of boundary layers tends to produce an acceleration similar to that observed in the experiments. It is proposed that this boundary layer induced interface acceleration leads to enhanced growth due to the Rayleigh-Taylor instability.
5:41PM EW.00008 Experimental Study of a Shock-Accelerated Gas Flow Non-Uniformly Seeded With Droplets, JOSEPH CONROY, MICHAEL ANDERSON, ROSS WHITE, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas-Brownsville — We present an experimental study of a gas flow which is partially seeded with a modest volume fraction of submicron-sized droplets and subjected to shock acceleration. Under these conditions an instability similar to Richtmyer-Meshkov develops. In our experiments, a planar shock front traveling horizontally through air meets a vertical column of gas (either air or SF6) that is seeded with particles. After shock interaction, the column is compressed and deformed, and a pair of counter-rotating vortices forms. The evolution of the flow is tracked with a multiple-CCD digital camera, allowing to capture up to four laser sheet-illuminated images per single experiment. We discuss the flow features in shocked and reshocked flows at a range of Mach numbers from 1.2 to 2.

1We thank Prof. S. “Bala” Balachandar (U. of Florida) for providing the suggestions that led to the experiments described here. This research is funded by NNSA through DOE Grant DE-PS52-08NA28920 and by DTRA awards HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

5:54PM EW.00009 Modeling a Shock-Accelerated Fluid - Multiphase Fluid Interface, MICHAEL ANDERSON, JOSEPH CONROY, C. RANDALL TRUMAN, PETER VOROBIEFF, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville — The hydrocode SHAMRC has been used in the past to study the formation and growth of the Richtmyer Meshkov Instability (RMI). While RMI involves impulsively accelerating two continuous fluids of differing densities, a similar class of instabilities has been recently described for multiphase flow. In this scenario, a shock wave passes through a region seeded with particles which have a non-trivial mass and density much greater than that of the surrounding and embedding fluid, resulting in a higher effective density in the seeded region. As the volume of the particles is small, there is no pressure gradient between the two regions. The simulations described here attempt to model the first order formation and growth phenomenon of this new class of instability by approximating the second phase as a continuous fluid with an averaged density. The strength of the shock and the packing density of the tracer particles are varied to provide a wide range of instability growth rates. Finally, these growth rates are scaled and compared to experimental data.

1This research is funded by NNSA through DOE Grant DE-PS52-08NA28920 and by DTRA awards HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

6:07PM EW.00010 Effect of multi-mode initial conditions in shock-driven flows, SRIDHAR BALASUBRAMANIAN, KATHERINE PRESTRIDGE, B.J. BALAKUMAR, GREGORY ORLICZ, GAVIN FRIEDMAN, LANL — Carefully imposed initial conditions have been shown to control late time turbulence and mixing in buoyancy-driven flows [Dimonte et al., 2004; Banerjee & Andrews, 2009]. This is important in understanding and prediction of Inertial Confinement Fusion. We report the experimental results on the initial condition parameters, namely amplitude (δ) and wavelength (λ) of perturbations, that impact the material mixing and transition to turbulence in shock-driven, Richtmyer-Meshkov instability. A detailed study on the impact of δ and λ on turbulence in a heavy gas varicose curtain (air-SF6-air) is undertaken. Experiments were conducted with stable, membrane-free initial conditions at shock Mach number, Ma = 1.2 and Atwood number, At = 0.67. The effect of multi-mode initial conditions on mixing and transition was quantitatively measured using simultaneous Particle Image Velocimetry (PIV) and Planar-Laser Induced Fluorescence (PLIF). The turbulence statistics were measured for different combinations of δ, λ. The results obtained are being compared with data from ongoing 3-D numerical simulations.

1Supported by LDRD-DR Program.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session EX Aerodynamics III Hyatt Regency Long Beach Regency D

4:10PM EX.00001 Time-dependent measurements over membrane plates at low Reynolds number, JAMES HUBNER, KYLE SCOTT, University of Alabama, AMORY TIMPE, LAWRENCE UKEILEY, University of Florida — A segment of low Reynolds number aerodynamic research employs biomimetics for optimization of airfoil shapes to micro air vehicle (MAV) flight. Many of these efforts focus on thin, flexible membrane airfoils inspired by small birds, bats and insects. This design approach, mimicking low Reynolds number flyers (Re < 100,000), has led to improved aerodynamic performance, particularly the mitigation of flow disturbances through passive aerodynamic and geometric twisting. In many cases, membrane vibration exists, altering the characteristics of the separated shear layer over the wing, leading to both advantageous and disadvantageous effects. Identifying and quantifying the nature of the fluid-structure coupling and how this coupling can passively control the flow is the goal of a recently initiated research project by the authors. This talk will present the objectives of the project and initial findings of synchronized flow (hot-wire anemometry) and surface deflection (laser vibrometry) measurements over rigid plates and flexible membranes at incidence to the free stream flow. A range of flow Reynolds numbers are examined, from 10,000 to 50,000, in which vibration initiates and grow with increasing velocity.

1Funded by AFOSR grant FA9550-10-1-0152; Dr. Doug Smith, Program Manager.

4:23PM EX.00002 Low dimensional state-space representations for classical unsteady aerodynamic models, STEVEN L. BRUNTON, CLARENCE W. ROWLEY, Princeton University — This work develops reduced order models for the unsteady aerodynamic forces on a small wing in response to agile maneuvers and gusts. In particular, the classical unsteady models of Wagner and Theodorsen are cast into a low-dimensional state-space framework. Low order state-space models are more computationally efficient than the classical formulations, and are well suited for modification with nonlinear dynamics and the application of control techniques. Reduced order models are obtained using the eigensystem realization algorithm on force data from the direct numerical simulation (DNS) of a pitching or plunging 2D flat plate at Reynolds numbers between 100 and 1000. Models are tested on rapid pitch and plunge maneuvers with a range of effective angle-of-attack. We evaluate the performance of the models based on agreement with results from DNS, in particular, the ability to reproduce lift forces over a range of pitching and plunging frequencies. Bode plots of the reduced order models, Wagner’s and Theodorsen’s methods, and DNS provide a concise assessment.

4:36PM EX.00003 The Dynamics of Spanwise Vorticity on a Rotating Blade in Unsteady Flow, CRAIG WOJCICKI, JAMES BUCHHOLZ, The University of Iowa — Spanwise flow driven by accelerations on rotating blades is known to influence the aerodynamic forces and moments in flapping flight and wind turbine aerodynamics compared with wings and airfoils that are stationary or oscillated in a planar motion. This difference is largely attributed to the resulting prolonged attachment of the leading edge vortex in the rotating case. In this experimental study, we consider the nature and dynamics of spanwise vorticity shed from the leading- and trailing edges of a spinning propeller that is yawed with respect to the free stream. Phase-averaged Digital Particle Image Velocimetry is used to interrogate the flow. The effects of tip speed ratio and yaw angle on development and strength of the shed structures will be discussed.

4:44PM EX.00004 Fluid-structure interaction in a hypersonic free-jet: a DNS study, THIAGO SEIXAS DE ARAUJO, JAMES K. PETERSON, THE UNIVERSITY OF WISCONSIN-MADISON, University Park, Pennsylvania — We present a study of a hypersonic jet, using direct numerical simulation (DNS) to investigate fluid-structure interactions. This approach, which resolves the full Navier-Stokes equations, allows for detailed characterization of the flow field, enabling an understanding of the complex dynamics that govern the interaction between the fluid and the structures. The DNS results provide insights into the aerodynamic forces and moments acting on the structures, and can be used to validate computational models. This study highlights the importance of fluid-structure interactions in hypersonic applications and their potential impact on design and performance.
constant-rate (linear) pitch, from angle of attack

equation, the LEV created during ramp-up motion remains over the foil to provide vortex lift at longer period, resulting in larger average lift over

during high-Frequency low-Re ramp and return are captured when \( k = 0.62 \). In addition, the spike in \( C_4 \) during high-Frequency low-Re ramp and return are captured when \( k = 0.62 \). The

drag at high \( \alpha \) and a rise in drag at high \( \alpha \). Noncirculatory and circulatory effects are additive and the noncirculatory portion is well predicted by potential-flow methods. \( K > 0.02 \) evinces the formation of a leading edge vortex. Peak in lift correlates to the angle of attack where the leading edge vortex reaches maximum circulation and begins to shed. Lift and drag are seen to obey a scaling with pitch rate, for \( K > 0.03 \).
M. Dietzel and S. M. Troian, Phys. Rev. Lett. cooler substrate as well as the warmer evolving viscous film can dominate intermolecular collisions in the gas film leading to temperature jump conditions first was assumed in these studies that the rate of heat transfer through the gas layer is well described by Fourier's law of thermal conduction. For sufficiently thin along the viscous film promote formation of fluid elongations resembling nanopillar arrays no matter how small the initial transverse temperature difference. It The bilayer consists of a warm molten viscous nanofilm overlay by a cooler nanofilm of quiescent gas. Lubrication analysis showed how thermocapillary stresses — We have previously conducted a linear stability analysis of a fluid bilayer system sandwiched in between flat solid substrates held at different temperature [1,2].

Confined Nanofilms

1 This work was partially funded by the DFG.

4:43PM EY.00002 Oscillatory surface patterns in a heated thin film of a binary mixture 1. MICHAEL BESTEHORN, BTU Cottbus, Germany — We study a thin liquid film with a free surface on a uniformly heated substrate. The film consists of a mixture of two arbitrarily miscible fluids. The surface tension depends both on temperature and on relative concentration of the mixture. Using lubrication approximation, a systematic derivation of the extended thin film equation is performed. The resulting coupled system of two conservation equations (mean thickness and mean concentration) is discussed by a linear stability analysis of the flat film. It turns out that for a large region in parameter space oscillatory (Hopf) instabilities are obtained which are long-wave. Numerical solutions of the non-linear problem finally show a rich spatio-temporal behavior of the film’s surface in form of traveling spots and holes or mazes having an intrinsic time dependence.

2 This research is supported by NSF (CBET-0651755) and NASA (NNX09AL02G).

5:02PM EY.00005 Influence of Gas Rarefaction on Thermocapillary Flow and Instability in Confined Nanofilms. NAN LIU, SANDRA TROIAN, California Institute of Technology, 1200 E California Blvd, MC 128-95, Pasadena, CA 91125 — We have previously conducted a linear stability analysis of a fluid bilayer system sandwiched in between flat solid substrates held at different temperature [1,2]. The bilayer consists of a warm molten viscous nanofilm overlay by a cooler nanofilm of quiescent gas. Lubrication analysis showed how thermocapillary stresses along the viscous film promote formation of fluid elongations resembling nanopillar arrays no matter how small the initial transverse temperature difference. It was assumed in these studies that the rate of heat transfer through the gas layer is well described by Fourier’s law of thermal conduction. For sufficiently thin gas layers, however, the mean free path for gas molecules is comparable to or can exceed the thickness of the gas layer. Collisions of the gas molecules with the cooler substrate as well as the warmer evolving viscous film can dominate intermolecular collisions in the gas film leading to temperature jump conditions first described by von Smoluchowski. In this presentation, we show how gas rarefaction effects can increase the growth rate and magnitude of the fastest growing wavelength by as much as 50% for parameter values relevant to experiment.

5:15PM EY.00006 Thermocapillary convection in a cylindrical liquid bridge – Effect of the ambient gas\(^1\), MICHAEL LUKASER, DANIEL LANZERSTORFER, HENDRIK KUHLMANN, Vienna University of Technology — The influence of the ambient gas phase on the stability of the two-dimensional axisymmetric steady flow in a cylindrical liquid bridge is investigated by a numerical linear stability analysis. The computational domain includes an annular gas channel which concentrically surrounds the liquid bridge. The stability boundaries strongly depend on the geometry and the material parameters. We consider a liquid bridge of \( \text{Pr} = 67 \) and focus on the effect of the gas-channel width and the thermophysical properties of the ambient gas. Stability boundaries, critical modes and mechanisms will be discussed.

\(^1\)This work has been supported by BMVIT under grant number 819714.

5:28PM EY.00007 Influence of adsorption kinetics on the Marangoni convection in a binary liquid layer with a soluble surfactant\(^1\), ALEXANDER A. NEPOMNYASHCHY, Technion – Israel Institute of Technology, Haifa, Israel, SERGEY SHKLYAEV, Institute of Continious Media Mechanics, Perm, Russia — We consider dynamics of a heated binary liquid in a horizontal layer. The solute is a soluble surfactant, which forms a surface phase and a bulk phase. Both the Soret effect and the adsorption kinetics are taken into account. Within the linear stability problem we demonstrate that the adsorption kinetics leads to a significant stabilization of the quiescent state with respect to oscillatory and monotonic longwave modes and a short-wave oscillatory mode. For the latter mode the stabilization is especially strong. Weakly-nonlinear analysis is carried out for the longwave modes, it results in an ill-posed set of amplitude equations. A subcritical bifurcation for both longwave modes is predicted, no nonlinear saturation takes place in the supercritical region. In a number of limiting cases more intricate amplitude equations are obtained; for the fast adsorption kinetics a transition between supercritical and subcritical bifurcations is found.

\(^1\)The work is supported by joint grants of the Israel Ministry of Sciences (Grant No. 3-5799) and the Russian Foundation for Basic Research (Grant No.09-01-92472) and by the European Union via FP7 Marie Curie scheme Grant PITN-GA-2008-214919 (MULTIFLOW).

5:41PM EY.00008 Thermocapillary fluid pumping using traveling thermal waves\(^1\), ALEX ON, Technion – IIT, WENBIN MAO, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use direct numerical simulations of the full Navier-Stokes and energy equations and the analysis based on the long-wave approximation to examine the dynamics of thin liquid films on substrates with periodic heating. Substrate temperature varies according to a sinusoidal law and creates periodic thermal waves that propagate unidirectionally along the substrate. Using the two modeling techniques, we probe how the traveling thermal waves can be harnessed to induce and regulate directed fluid flows in the liquid film. We examine emerging flow structures, film deformation, and characterize the fluid flow in terms of relevant dimensionless parameters. Furthermore, we identify the optimal parameters leading to the efficient fluid pumping. The results of our studies can be useful for designing open microfluidic devices, in which the thermocapillary pumping is employed for controlled transport of samples in free-surface liquid films.

\(^1\)Support from the US-Israel Binational Science Foundation (Grant No. 2008038), the European Union via the FP7 Marie Curie scheme [PITN-GA-2008-214919 (MULTIFLOW)], and NSF (CBET-1028778) is gratefully acknowledged.

5:54PM EY.00009 On the unsteady Benard – Marangoni problem, ITZCHAK FRANKEL, MICHAEL WIDENFELD, Technion - Israel Institute of Technology — The Benard – Marangoni instability originating from the coupling between a non-uniform liquid – surface temperature and convection across a favorable temperature gradient is a classic problem in hydrodynamic stability theory. We study the linear temporal stability problem focusing on short times when the modes evolve on the background of a non-linear unsteady temperature distribution across the liquid layer. Accordingly, we analyze the initial – value (rather than the standard eigenvalue) problem and obtain a Volterra – type integral equation governing the evolution of perturbations. The results indicate that the onset of convection at short times is effectively confined to the narrow thermal boundary layer developing next to the liquid surface and is dominated by short – wavelength perturbations. Increasing the Prandtl number is found to be destabilizing.

Sunday, November 21, 2010 4:10PM - 6:20PM – Session EZ Couette and Swirling Flows

4:10PM EY.00001 Subcritical Transition and Spiral Turbulence in Taylor-Couette Flow, M.J. BURIN, C.J. CZARNOCKI, T. DAPRON, K.R. MCDONALD, CSU San Marcos — We present measurements characterizing the transition to turbulence in Taylor-Couette flow for a fully cyclonic regime, i.e. with only the outer cylinder rotating. Under this arrangement the flow is linearly-stable and the shear-driven transition to turbulence is understood to be both ‘catastrophic’ and spatiotemporally intermittent. En route to a fully turbulent state, we observe a regime featuring co-extant laminar/turbulent domains known as spiral turbulence. To better understand this regime, and the transition in general, we have obtained velocimetry data (via LDV) and angular momentum transport estimates (via torque), in addition to flow visualization. These observations are discussed with respect to similar transition phenomena in planar and counter-rotating Couette flows. By utilizing three different inner cylinder radii within the apparatus, we also demonstrate the sensitivity of the subcritical transition scenario to annular gap width.

4:23PM EY.00002 Torque scaling in turbulent Taylor-Couette flow with independently rotating cylinders, MATTHEW S. PAOLETTI, University of Texas at Austin, DANIEL P. LATHROP, University of Maryland at College Park — We present experimental studies of the turbulent flow of water between independently rotating cylinders. The Taylor-Couette system is capable of both strong turbulence (\( Re > 2 \times 10^9 \)) and rapid rotation. The torque required to drive the inner cylinder and the wall shear stress at the outer boundary are precisely measured as a function of the two angular velocities \( \Omega_1 \) and \( \Omega_2 \). We find that the dynamics, which are fully determined by the Reynolds number \( Re \) and Rossby number \( Ro = \Omega_1 - \Omega_2 / \Omega_2 \), are different in four different regions of the \( (\Omega_1, \Omega_2) \) parameter space. Our measurements allow us to estimate the skin friction coefficient \( c_f \). We compare our measurements of \( c_f \) with those of previous experiments and discuss the potential relevance for angular momentum transport in astrophysical flows.

4:36PM EY.00003 Turbulent scaling in rotating spherical Couette flow, DANIEL S. ZIMMERMAN, SANTIAGO ANDRES TRIANA, University of Maryland Physics/IREAP, DANIEL P. LATHROP, University of Maryland Physics/IREAP/Geology — We study the parameter dependence of torque and other flow quantities in rapidly rotating spherical Couette flow with radius ratio \( r_1/r_0 = 0.35 \) using the University of Maryland 3m system. We examine the dependence of the dimensionless torque, \( G = T/\rho u^2 L_{gap} \), on the Reynolds number, \( Re = \Delta \Omega L_{gap}^2/\nu \), and Rossby number, \( Ro = \Delta \Omega / \Omega_0 \), for \(-5 < Ro < 90 \) and \( 5 \times 10^5 < Re < 2 \times 10^7 \). In this range, \( G \) is described well as a power law in \( Re \) multiplied by a \( Ro \)-dependent prefactor: \( G \propto f(Re)Ro^{1.9} \). The turbulent flow exhibits several distinct transitions as \( Ro \) is varied; some of these exhibit bistability between adjacent states with significantly different torque demand. In the bistable ranges, the prefactor \( f(Re) \) is multi-valued. The complicated dependence on \( Ro \) and the simple dependence on \( Re \) may have important consequences for the prediction of turbulent transport in rapidly rotating, strongly turbulent flows like those found in planetary cores, oceans, and other geophysical and astrophysical objects.
4:49PM EZ.00004 Taylor–Couette–Poiseuille flow with a permeable inner cylinder1, NIFS TILTON, DENIS MARTINAND, Universités Aix-Marseille, ERIC SERRE, CNRS, RICHARD LUEPTOW, Northwestern University — We consider laminar Taylor–Couette–Poiseuille flow between an outer, fixed, impermeable cylinder and a concentric, inner, rotating, permeable cylinder with radial suction. Due to centrifugal instabilities the steady flow transitions to Taylor vortex flow. This system is used in filtration because the vortices wash contaminants away from the permeable cylinder. The coupling between the axial pressure drop driving the annular Poiseuille flow, and the transmembrane pressure driving the suction induces axial variations of the velocity field of the subcritical flow, which can evolve from suction to injection (cross flow reversal) or consume the whole axial flow (axial flow exhaustion). Moreover, the stability of this flow depends on that of Taylor–Couette flow. We propose an asymptotic solution to the subcritical flow assuming slow axial variations of the velocity and pressure fields. The transmembrane suction and pressure are coupled through Darcy’s law. This solution is then used as a base flow to study the appearance of instabilities in the form of global modes. The analytical results for the subcritical and supercritical flows are then compared with dedicated 3-D spectral direct numerical simulations implementing Darcy’s law on the inner cylinder.

3Work supported by ANR program No. ANR-08-BLAN-0184-03.

5:02PM EZ.00005 Precessional states in a laboratory model of the Earth’s core1, SANTIAGO TRIANA, DANIEL ZIMMERMAN, DANIEL LATHROP, IREP/ University of Maryland — A water-filled three-meter diameter spherical shell built as a model of the Earth’s core shows evidence of precessional induced flows. We identified the flow to be primarily the spin-over inertial wave mode, i.e., a uniform vorticity flow whose rotation axis is not aligned with the container’s rotation axis. The mode’s amplitude dependence on the Poincaré number is in qualitative agreement with Busse’s laminar theory (JFM 33:739-751, 1968) while its phase differs significantly, perhaps due to topographic effects. At high rotation rates free shear layers concentrating most of the kinetic energy of the mode have been observed. Comparison with previous computational studies and implications for the Earth’s core are discussed.

1Support from NSF Earth Sciences is greatly acknowledged.

5:15PM EZ.00006 Enhanced Angular Momentum Transport in Magnetized Spherical Couette Flow, MATTHEW ADAMS, DANIEL LATHROP, University of Maryland, College Park — We present experimental studies of the turbulent flow of a conducting fluid in a spherical shear flow in the presence of a magnetic field. Our experimental apparatus uses sodium as the working fluid, and both the inner and outer spheres can be rotated independently. An axial magnetic field of varying strength can be applied to the experiment, and magnetic field measurements are used to extract information about the global flow within the device. In addition, we measure the torque required to drive the inner and outer spheres at their respective rotation rates. The geometry of the experiment makes these studies applicable to geophysical and astrophysical bodies. With the inner sphere rotating faster than the outer sphere, we observe enhanced angular momentum transport from the inner to the outer sphere as the applied magnetic field is increased. In an experiment of the same geometry, enhanced angular momentum transport was observed with a stationary outer sphere [1]. In this case the source of enhanced transport was identified as the magnetorotational instability. Results for the case of rotating outer sphere also indicate the possible presence of the magnetorotational instability with independently rotating spheres.


5:28PM EZ.00007 Drag Measurements over Embedded Cavities in a Low Reynolds Number Couette Flow1, CALEB GILMER, University of Maryland, Baltimore County, AMY LANG, ROBERT JONES, University of Alabama — Recent research has revealed that thin-walled, embedded cavities in low Reynolds number flow have the potential to reduce the net viscous drag force acting on the surface. This reduction is due to the formation of embedded vortices allowing the outer flow to pass over the surface via a roller bearing effect. It is also hypothesized that the scales found on butterfly wings may act in a similar manner to reduce a net increase in flying efficiency. In this experimental study, rectangular embedded cavities were designed as a means of successfully reducing the net drag across surfaces in a low Reynolds number flow. A Couette flow was generated via a rotating conveyor belt immersed in a tank of high viscosity mineral oil above which the plates with embedded cavities were placed. Drag induced on the plate models was measured using a force gauge and compared directly to measurements acquired over a flat plate. Various cavity aspect ratios and gap heights were tested in order to determine the conditions under which the greatest drag reductions occurred.

1Work performed under REU site sponsored by NSF grant EEC 0754117.

5:41PM EZ.00008 Computational Analysis of Low Reynolds Number Couette Flow Over Embedded Cavities1, CHASE LEIBENGUTH, AMY LANG, WILL SCHREIBER, University of Alabama — Bio-inspired surface patterning research has shown the potential drag reduction qualities of micro-geometric embedded cavities placed on the surface of an object, analogous to the spaces formed between successive rows of scales on a butterfly wing. Vortices form inside the cavities and contribute to a net partial slip condition that interacts with the boundary layer over the surface. The interaction potentially affects the global flow field over an object to delay separation and reduce drag. In the present study, embedded cavity geometry in a Couette flow was modeled in GAMBIT and analyzed with FLUENT to qualitatively determine the cavity’s drag reduction capabilities and the presence of a partial slip condition. The GAMBIT models consisted of a top plate moving transversely over a single cavity with periodic boundary conditions, differing rectangular geometry configurations, and varied gap heights. FLUENT was used to analyze the flow over a range of Reynolds Numbers from 0.01 to 100. Data was obtained to analyze cavity vortex formation, pressure and shear distributions inside the cavity, and the velocity distribution near the cavity.

1Work performed under REU site sponsored by NSF grant EEC 0754117.

5:54PM EZ.00009 Slow dynamics in a highly turbulent von Kármán swirling flow, MIGUEL LOPEZ, JAVIER BURGÜETE — In this work we present an experimental analysis of the dynamics of the coherent structures that appear in a von Kármán swirling flow, in a fully developed turbulent regime. The objective is to determine the effect of the fluctuations in the dynamics of these vortices. To achieve this goal, we have measured the flow in a water experiment. The fluid has been stirred in a cylindrical environment up to a Reynolds number of 10^6. We show that the average velocity field of the turbulent flow bifurcates subcritically breaking some symmetries of the problem and becomes time-dependent because of equatorial vortex moving with a precession movement. This subcriticality produces a bistable regime, with a hysteresis region for an extremely small range of parameters. Three different time-scales are relevant to the dynamics, two of them very slow compared to the impeller frequency. We have studied the different time scales of the system, changing an enclosure volume (neutrally buoyant spheres) assuming that the density of the sphere is homogeneous. Also we change the frequency of the impellers (10Hz - 50Hz) to explore another parameter of the system. We follow this volume in a period of time and we compare the results in different spatial scales.
Lagrangian time evolution of velocity gradient dynamics near the Vieillefosse tail. The data are obtained from fluid particle tracking through the turbulence near the Vieillefosse tail and provides its survival against turbulent mixing. A simple experiment is performed confirming the counterflow geometry.

Pressure drops from the inlet to the dead end near the sidewall and from the dead end to the exhaust near the axis. Such a pressure distribution drives the counterflow as the inlet and exhaust are close, there is no short-cut flow. The fluid goes from the inlet near the sidewall to the dead end, turns around, and goes back near the axis. This global counterflow occurs due to swirl decay caused by friction at the sidewall. The combined effect of swirl and friction causes that pressure drops from the inlet to the dead end near the sidewall and from the dead end to the exhaust near the axis. Such a pressure distribution drives the counterflow and provides its survival against turbulent mixing. A simple experiment is performed confirming the counterflow geometry.

— Swirling counterflows occur in vortex combustors, hydrocyclones, and vortex tubes where the Reynolds number can exceed million. It is explained here why.

The present is the Taylor micro length scale. According to the analysis of one of the layers, coarse grained vorticity in the layer aligns roughly in one direction, large velocity jump of order of magnitude as large as almost the root-mean-square of the fluctuating velocity occurs across the layer, and energy dissipation averaged over the layer is larger than ten times the average over the whole flow. The mean and the standard deviation of the energy transfer \( T(k_x) \) from scales larger than \( 1/k \) to scales smaller than \( 1/k \), at position \( x \) in the layer are larger than those outside the layer, but the probability distribution function of \( T \) in the layer under an appropriate normalization is similar to that outside the layer. The fact that the correlation of velocity fluctuation falls sharply at one of the boundary of the layer suggests that the boundary acts as a barrier of turbulent fluctuations. The space filliness of such shear layers will be also discussed in the talk.

---

Monday, November 22, 2010 8:00AM - 10:10AM —
Session GA Turbulence Theory I
Long Beach Convention Center 101A

8:00AM GA.00001 Public-database enabled analysis of Lagrangian dynamics of isotropic turbulence near the Vieillefosse tail

— HUIDAN YU, CHARLES MENEVEAU, Johns Hopkins University, Baltimore, MD — We study the Lagrangian time evolution of velocity gradient dynamics near the Vieillefosse tail. The data are obtained from fluid particle tracking through the 1024 space-time DNS of forced isotropic turbulence at \( Re_k = 433 \), using a web-based public database (http://turbulence.pha.jhu.edu). Examination of individual time-series of velocity gradient invariants \( R \) and \( Q \) show that they are punctuated by strong peaks of negative \( Q \) and positive \( R \). Most of these occur very close to the Vieillefosse tail along \( Q = -(3/2^{2/3})Q^2/k^2 \). It is found there that the magnitude of pressure Hessian has positive Lagrangian time-derivative, meaning that it increases in order to resist the rapid growth. We also observe a “phase delay” of the pressure Hessian signals compared to those of \( R \) and \( Q \), indicative of an “overshoot” of the controlling mechanism. We also examine the trajectories in the recently proposed 3-D extension of the \( R - Q \) plane (see Lüthi B, Holzner M, Tsinober A., 2009, J. Fluid Mech. 641, 497-507). Finally, Lagrangian models of the velocity gradient tensor are examined in the same light to identify similarities and differences with the observed dynamics. Such comparisons supply informative guidance to model improvements.

8:13AM GA.00002 Signatures of non-universal large scales in conditional structure functions from eight different turbulent flows

— GREG VOTH, DANIEL BLUM, Wesleyan University, EBERHARD BODENSCHATZ, MATHIEU GILBERT, HAITAO XU, MPI Goettingen, LAURENT MYDLARSKI, McGill University, ARMANM GYLFASON, Reykjavik University, P.K. YEUNG, Georgia Tech — We present a systematic comparison of conditional structure functions in eight turbulent flows. The flows studied include DNS of a periodic box, passive grid wind tunnel, active grid wind tunnel (in both synchronous and random driving modes), counter-rotating disks, oscillating grids, and the Lagrangian exploration module (in both constant and random driving modes). We compare longitudinal Eulerian second order structure functions conditioned on the instantaneous large scale velocity in order to assess ways in which the large scales affect the small scales in a wide variety of turbulent flows. Structure functions are shown to have larger values when the large scale velocity is large in all flows except the passive grid wind tunnel and DNS indicating that dependence on the large scales is typical in turbulent flows. The large scales can be quite significant, with the structure function varying by up to a factor of 2 when the large scale velocity changes by 2 standard deviations. In general, the conditional dependence of the structure functions on the large scale velocity is similar at all scales which indicates large scale effects are scale independent.

8:26AM GA.00003 Conditional statistics near strong thin shear layers in DNS of isotropic turbulence at high Reynolds number

— TAKASHI ISIHARA, Nagoya University, JULIAN C.R. HUNT, University College London, YUKIO KANEDA, Nagoya University — Data analysis of high resolution direct numerical simulations (DNS) of isotropic turbulence with the Taylor scale Reynolds number up to 1131 shows that there are thin shear layers consisting of a cluster of strong vortex tubes. The widths of the layers are approximately 5\( \lambda \), where \( \lambda \) is the Taylor micro length scale. According to the analysis of one of the layers, coarse grained vorticity in the layer aligns roughly in one direction, large velocity jump of order of magnitude as large as almost the root-mean-square of the fluctuating velocity occurs across the layer, and energy dissipation averaged over the layer is larger than ten times the average over the whole flow. The mean and the standard deviation of the energy transfer \( T(k_x) \) from scales larger than \( 1/k \) to scales smaller than \( 1/k \), at position \( x \) in the layer are larger than those outside the layer, but the probability distribution function of \( T \) in the layer under an appropriate normalization is similar to that outside the layer. The fact that the correlation of velocity fluctuation falls sharply at one of the boundary of the layer suggests that the boundary acts as a barrier of turbulent fluctuations. The space filliness of such shear layers will be also discussed in the talk.

8:39AM GA.00004 Decay of fractal-generated homogeneous turbulence

— PEDRO VALENTE, CHRISTOS VASSILICOS, Imperial College London — We present new hot wire anemometry measurements of decaying homogeneous, quasi-isotropic turbulence generated by low-blockage space-filling fractal square grids using different anemometry systems and hot-wires of decreasing diameter for increased spatial resolution. We find good agreement with previous works by Seoud & Vassilicos (2007) and Mazellier & Vassilicos (2010) but also extend the length of the assessed decay region. It is shown that the measured 1D spectra can be reasonably collapsed using a single length-scale (George 1992, George & Wang 2009) over the entire decay region even though the Reynolds number is high enough for conventional decaying turbulence to display 1D spectra with two-scale (inner and outer) Kolmogorov scaling. The weak anisotropy of the flow can be accounted for by computing the 3D spectrum function from two component velocity signals leading to further improved single-scale non-Kolmogorov collapse. Detailed checks on homogeneity and isotropy are presented as well as measurements with a regular grid indicating that the single-length scale locking is neither an artifice of, hardly present, inhomogeneity nor an effect of confinement from the wind-tunnel walls.

---

1Supported by NSF (CDI-II, CMMI-0941530).

2Funded by the ONR contract # N0001409C0121.
8:52AM GA.00005 Asymptotic analysis of homogeneous isotropic decaying turbulence with unknown initial conditions. PHILIP SCHAEFER, MARKUS GAMPERT, JENS HENRIK GOEBBERT, MICHAEL GAUDING, PETERS NORBERT, Institute for Combustion Technology, RWTH Aachen — In decaying grid turbulence there is a transition from the initial state immediately behind the grid to the state of fully developed turbulence downstream which is believed to become self-similar and is characterized by a power law decay of the turbulent kinetic energy with a decay exponent. The value of this exponent however depends on the initial distribution of the velocity moments. In the non-dimensionalized form of the von Kármán-Howarth equation a decay exponent dependant term occurs whose coefficient will be called δ. We exploit the fact that δ vanishes for ν = 2 to formulate a singular perturbation problem, where another small number in the equation, namely 1/4, is assumed to be of the same order as of magnitude as δ. In the limit of infinitely large Reynolds numbers, we obtain an outer layer as well as an inner layer of the thickness of the order O(δ^2), where the Kolmogorov scaling is valid. To leading order, we obtain in the outer layer an algebraic balance between the two-point correlation and the third order structure function.

9:05AM GA.00006 Scaling of the two-point velocity difference along scalar gradient trajectories. MARKUS GAMPERT, PHILIP SCHAEFER, JENS HENRIK GOEBBERT, NORBERT PETERS, Institute for Combustion Technology, RWTH Aachen — To analyze the geometrical properties of scalar turbulent fields, the concept of dissipation elements has been proposed by Peters and Wang (J. Fluid Mech. 2006, 2008). Starting from every grid point, trajectories can be traced in directions of ascending and descending gradient until a local extreme point is reached. Based on these trajectories, a dissipation element is defined as the region containing all grid points, whose trajectories share the same pair of extreme points. To parameterize dissipation elements, the linear length between and the scalar difference at the extreme points have been chosen. While the conditional scalar difference follows Kolmogorov scaling, Wang (Phys. Rev. E, 2009) showed that the velocity difference between maximum and minimum follows a linear scaling proportional to τ/λ, where τ is the integral time scale and λ the Taylor microscale. In this context, the intention of the present paper is the comparison and analysis of the scaling of the conditional velocity difference in the viscous and the inertial range. Therefore, Direct Numerical Simulations (DNS) of different turbulent flows at various Reynolds numbers Re = 70 – 300 are studied and discussed. It is concluded that the scaling is valid in all the above mentioned flows and thus poses a universal character.

9:18AM GA.00007 DNS and Rapid Distortion Theory investigations of Mach number effects on velocity-pressure field interactions in strongly sheared flows. GAURAV KUMAR, SHARATH GIRIMAJI, REBECCA BERTSCH, Texas A&M University — Computations based on the Rapid Distortion Theory of homogeneously sheared compressible flow show three distinct types of velocity and thermodynamic field interactions depending upon the gradient Mach number. The difference arises due to the changing role of pressure at sub-sonic, sonic and hypersonic gradient Mach numbers. To understand and further examine the varying role of pressure, we perform direct numerical simulations (DNS) of compressible homogeneous shear flow using Gas Kinetic Method (GKM). We investigate linear and non-linear effects of a) gradient Mach number and b) perturbation Mach number and c) initially thermodynamic fluctuations on velocity-thermodynamics interactions. This study is expected to contribute toward the development of improved transition and turbulence closure models in highly compressible shear flows.

9:31AM GA.00008 Effects of Mach number and compressibility on vorticity and strain-rate turbulence dynamics. SAWAN SUMAN, SHARATH GIRIMAJI, Texas A&M University — We study the effects of Mach number and compressibility on strain-rate and vorticity dynamics in decaying isotropic turbulence employing direct numerical simulations. Since local Mach number and dilatation are two direct indicators of compressibility of a fluid element, we use these quantities as conditioning parameters to examine the various aspects of turbulence dynamics. Several interesting observations along with the underlying physics pertaining to the inertial (vortex stretching and self-straining) and pressure (pressure Hessian and baroclinic) terms in the budget of strain-rate and vorticity dynamics will be presented in the talk. The contrasting nature of these physical effects in expanding vs. contracting and supersonic vs. subsonic fluid elements will be highlighted.

9:44AM GA.00009 Compressible Turbulence: Cascade, Locality, and Scaling. HUSSEIN ALUIE, Los Alamos National Laboratory — While Kolmogorov’s 1941 phenomenology forms the cornerstone for our understanding of incompressible turbulence, no analogous results exist for compressible flows. We present a rigorous framework to analyzing highly compressible turbulence. We show how the sole requirement that viscous effects on the dynamics of large-scale flow be negligible naturally leads to a density weighted coarse-graining of the velocity field, also known as Favre averaging. We prove that there exists a range of scales over which viscous and large-scale forcing contributions are negligible in the kinetic energy budget. An important part of our work proves that the non-linear transfer of kinetic energy to small scales is in the form of a local cascade process. Using scale-locality, we show that the average pressure-dilatation only acts at large-scales and that the mean kinetic and internal energy budgets statistically decouple beyond a “conversion” scale-range. We rigorously prove that over the ensuing inertial range, scaling exponents of velocity structure functions (⟨|du|⟩^{1/p} ≈ ℓ_p) are constrained by 1/3 ≥ σ_ρ for all p ≥ 3. By assuming self-similarity, we show semi-rigorously that σ_p = 1/3 for p > 0 which implies a Kolmogorov spectrum E_p(k) ≈ k^{−5/3} for the velocity field.

9:57AM GA.00010 Life at high Reynolds number. PRASAD PERLEKAR, Department of Physics, and Department of Mathematics and Computer Science, Eindhoven University of Technology, Eindhoven, ROBERTO BENZI, Dip. di Fisica and INFN, Universita Tor Vergata, Rome, DAVID NELSON, Lyman Laboratory of Physics, Harvard University, Cambridge, USA, FEDERICO TOSCHI, Department of Physics, and Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands — We study the statistical properties of population dynamics evolving in a realistic two-dimensional compressible turbulent velocity field: mimicking a surface flow. We show that the interplay between turbulent dynamics and population growth leads to quasi-localization and a remarkable reduction in the carrying capacity. The statistical properties of the population density are investigated and quantified via multifractal analysis. We investigate numerically the limit of negligibly small growth rates and delocalization of population ridges triggered by uniform advection. We also study the role of compressibility on the quasi-localization.

Monday, November 22, 2010 8:00AM - 10:10AM
Session GB Turbulent Boundary Layers IV  Long Beach Convention Center 101B
8:00AM GB.00001 Modeling roughness effects in turbulent boundary layers using elliptic relaxation
[1] JACOB GEORGE, Innovative Aerospace Solutions, Downey, CA, ALEJANDRO DE SIMONE, GIANLUCA IACCARINO, Center for Turbulence Research (CTR), Stanford University, Stanford, CA, JAVIER JIMENEZ, School of Aeronautics, U. Politecnica, Madrid, Spain, also CTR, Stanford — We present results from the efforts toward modeling roughness in turbulent boundary layers using elliptic relaxation. This scheme, included in the $c^2- f$ model and first formulated by Durbin (1993, JFM, vol. 249, p.465) for smooth-walls, uses an elliptic partial differential equation to incorporate near-wall turbulence anisotropy and non-local pressure-strain effects. The use of the elliptic PDE is extended to model roughness effects in various transitionally-rough and fully-rough boundary layers consisting of a uniform and sparse distribution of cylinders for which experimental data is available. The roughness effects are incorporated through the elliptic PDE by including the length and time-scales that the roughness imposes upon the flow, which the experiment has shown to be constant within the rough-walls. Further modeling of roughness effects is considered by altering the source terms in the elliptic PDE.


8:13AM GB.00002 DNS of a Turbulent Boundary Layer with Surface Roughness
[1] YI CHEN, JAMES CARDILLO, Rensselaer Polytechnic Institute, GUILLERMO ARAYA, Swansea University, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, KENNETH JANSEN, University of Colorado Boulder — A Direct numerical simulation (DNS) of a high Reynolds number, zero pressure gradient, turbulent boundary layer ($Re_\theta = 2400$) subjected to sandpaper surface roughness is performed. The surface roughness is modeled with a roughness parameter $k^+ \sim 25$ to match the experiments at similar Reynolds number and roughness distribution. The employed computational method involves a synergy of the multi-scale dynamic approach devised by Araya et al. (2010) and a new method for mapping high-resolution topographical data onto a computational domain. When dealing with rough surfaces, where calculation of the wall shear stress is very challenging the multi-scale dynamic method provides a major advantage. Contrary to traditional thought, it has been shown that the different types of surface roughness yield different types of flow fields. In light of these challenges the current roughness methodology aims to provide the community with the tools to use real topographical data to simulate surface roughness. The present simulations may shed light on our understanding of the interaction of the outer and inner layers at various scales.

[2] Supported by NSF (CBET- 0839209) and ONR (Dr. R. Joslin).

8:26AM GB.00003 Inner and Outer Flow Interactions on Rough, Turbulent Boundary Layers
[1] SHEILLA TORRES-NIEVES, Rensselaer Polytechnic Institute, HYUNG-SUK KANG, CHARLES MENEVEAU, The Johns Hopkins University, LUCIANO CASTILLO, Rensselaer Polytechnic Institute — Laser Doppler and hotwire anemometry measurements are performed to study the effects of surface roughness on the different length scales of the turbulent boundary layer. Measurements are carried out downstream of an active grid, with free-stream turbulence and Reynolds number, based on momentum thickness, of up to 6% and 4,300, respectively. Second-order structure functions and energy spectra distributions are used to identify and examine how surface roughness affects the inner and outer regions of the boundary layer. Second order structure function analysis suggests that, for favorable pressure gradient flows, surface roughness directly interacts with, not only the small length scales of the flow, but also with intermediate and even large scales. These observations are even more evident when additional levels of turbulence are present in the free-stream. Power spectral density plots are analyzed in order to understand the mechanism by which larger length scales interact with the surface roughness at the wall.

8:39AM GB.00004 Combined Roughness and Favorable Pressure Gradient Effects in a Turbulent Boundary Layer, D. MIN, K.T. CHRISTENSEN, U. Illinois — The combined impact of irregular surface roughness and moderate favorable-pressure-gradient (FPG) conditions ($K \approx 2.5 \times 10^{-3}$) on the structure of a turbulent boundary layer is assessed using two-dimensional particle image velocimetry (PIV) measurements in the streamwise–wall-normal plane. The roughness under consideration is replicated from a turbine blade damaged by deposition of foreign materials and contains a broad range of topographical scales. These measurements are compared to measurements of smooth-wall flow under both identical FPG conditions as well as zero-pressure-gradient (ZPG) conditions in order to reveal the synergistic impact of roughness and FPG conditions on the underlying structure of the flow. While the organization of the flow is found to persist under both smooth- and rough-wall FPG conditions, its characteristics are altered compared to smooth-wall ZPG flow. Inspection of instantaneous velocity fields reveals that this organization is to be focused closer to the wall in the smooth-wall FPG case, with a shallower inclination angle noted as well as an elongated streamwise extent. In contrast, the rough-wall FPG results reveal packet structures more consistent with the smooth-wall ZPG case, indicating that roughness mitigates the FPG-induced focusing of these structural attributes toward the wall. Two-point correlations of streamwise velocity highlight the importance of understanding the mechanism by which larger length scales interact with the surface roughness at the wall.

8:52AM GB.00005 Studies of the Combined Effects of Roughness and Reynolds Number in Turbulent Boundary Layers[1], FARAZ MEHDI, JOSEPH KLEWICKI, University of New Hampshire — Mehdi, Klewicki & White [Physica D 239(2010)] provide evidence from existing studies that the prevalent scheme for classifying roughness regimes is likely to be incomplete. To further pursue these findings, more data are required, and for this purpose, additional rough-wall experiments are being performed. We report on our studies of the combined roughness-Reynolds number problem conducted in a 8m long wind-tunnel. The roughness considered is the randomly distributed type and introduced in the form of 24-grit sandpaper and pea gravel. The primary measurement tool is two-component LDV. The basis of the analysis is the mean equation of dynamics. In this regard, the length scale defining where the mean dynamics become dominated by inertia is of central importance.

[1] The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.

9:05AM GB.00006 Structural Aspects of Wall Turbulence Over Low-Order Representations of Irregular Roughness, R. MEJIA-ALVAREZ, K.T. CHRISTENSEN, U. Illinois — Low-order representations of roughness replicated from a turbine blade damaged by deposition of foreign materials are generated using singular value decomposition (SVD) to decompose the surface into a set of topographical basis functions (383 total) of decreasing importance to the original (“full”) surface character. The low-order surface models are then formed by truncating the full set of basis functions at the first 5 and 16 modes (containing approximately 71% and 95% of the full surface content, respectively), so that only the most dominant, and large-scale, topographical features are included in the models. Physical replications of the full surface and the two models are created by rapid prototyping and PIV is used to measure velocity fields for all cases in both wall-normal and wall-parallel planes from which the structural aspects of these flows are explored. While the three rough-wall flows are found to be similar to smooth-wall flow outside the roughness sublayer, differences are noted within the roughness sublayer. Persistent low- (LMR) and high-momentum (HMR) regions often bounded by regions of enhanced Reynolds stresses are observed at preferential spanwise positions that vary depending upon the details of the topography. These observations suggest the possibility of enhanced production and dissipation at preferential locations within the roughness sublayer that depend upon the details of the topography.
Incompressible flow around a small obstacle and the vanishing viscosity limit, see [1], ongoing research on homogenization, and open problems. The description of fluid-solid interaction at large Reynolds number. In this talk, we will present the main results obtained, focusing especially on the joint small incompressible flow around small obstacles, both in the inviscid and viscous cases. These results showcase the difficulties and mathematical issues surrounding explicit roughness drag term that arises due to this formulation.

1. Supported under ONR Grant N00014-09-1-0263, monitored by Ron Joslin.

FLUIDS GROUP COLLABORATION — In recent years, the author and his research team have obtained several results concerning the limiting behavior of \( \text{Re}_T \) for \( \text{Re}_T = 0.09 \). The value of \( k / \delta \) is greater than 350 in both cases indicating that the flow is in the fully-rough regime. The Reynolds stresses appears to exhibit outer-layer similarity for \( \text{Re}_T = 1800 \) and 4000. The results indicate that the relative roughness height of these elements \( (k / \delta) \) is approximately 0.09. The value of \( k_s \) is greater than 350 in both cases indicating that the flow is in the fully-rough regime. The mean velocity profile in defect form at both Reynolds numbers conforms to outer-layer similarity. The Reynolds stresses appears to exhibit outer-layer similarity for \( y / \delta = 0.12 \) for \( \text{Re}_T = 4000 \). Further analysis using both the PIV and LDV data will be performed and presented.

2. Support from the Office of Naval Research
8:39AM GC.00004 Variance and Skewness Evolution in Transient Taylor Dispersion, KEITH MERTENS, ROBERTO CAMASSA, RICH MCLAUGHLIN, University of North Carolina, Mathematics, NICK MOORE. Courant Institute of Mathematical Sciences, MATT HERNANDEZ, WILLIAM MILLIKEN, University of North Carolina, Mathematics — This talk will report on a combined numerical, experimental, and theoretical study of the variance and skewness evolution for passive scalar particles transported in pipe and channel flows, with explicit differences between these two cases illustrated and explained. We will investigate the dependence of initial conditions and Peclet number in effecting evolution dynamics within the first diffusive timescale. Questions concerning how the properties of a given initial condition determine the time required for asymptotic theory to become valid will also be addressed.

1NSF

8:52AM GC.00005 Passive scalar advection in parallel shear flows: WKBJ mode sorting on intermediate times, RICHARD MCLAUGHLIN, ROBERTO CAMASSA, CLAUDIO VIOTTI, University of North Carolina, NSF UNC RTG FLUIDS GROUP COLLABORATION — The evolution of a passive scalar diffusing in simple parallel shear flows is a problem with a long history. In 1953, GI Taylor showed theoretically and experimentally that on long times, the passive scalar experiences an enhanced diffusion in the longitudinal direction. On shorter times the scalar evolution is anomalous, characterized by second moments growing faster than linear in time as we show by analysis of the stochastic differential equations underlying the passive scalar equation. The spatial structures associated with this intermediate time evolution are predicted using WKBJ analysis of an associated non-self adjoint eigenvalue problem. This analysis predicts a sorting of wall modes and interior modes with specific predictions of the decay and propagation rates as a function of the Peclet number.

1NSF

9:05AM GC.00006 Structure from the critical layer framework in turbulent flow, ATI SHARMA, Imperial College, BEVERLEY MCKEON, California Institute of Technology — We extend the critical layer framework for turbulent pipe flow proposed by McKeon & Sharma (J. Fluid Mech, 2010) to investigate vortical structure generated at particular streamwise/azimuthal wavenumber and frequency combinations, (k, n, ω). This framework utilizes an input-output formulation of the Navier-Stokes equations in a divergence-free basis to analyze the transfer function (the “resolvent”) and identify the dominant forcing and response mode shapes at each (k, n, ω) combination relevant to experimental spectra. It is shown that the hairpin vortex is a natural constituent of the velocity field associated with so-called wall modes, such that our model gives important predictive information about both the statistical and structural make-up of wall turbulence. Thus the dominant response mode shapes form a suitable basis by which to decompose the full turbulent velocity field. Acknowledgements: This research is sponsored by an Imperial College Junior Research Fellowship and the AFOSR (program manager J. Schissmee).

9:18AM GC.00007 A topological approach to three-dimensional laminar mixing, MATTHEW FINN, NATHANIEL JEWELL, The University of Adelaide — Research into laminar mixing has enjoyed a renaissance in the last decade since the realisation that the Thurston–Nielsen (TN) theory of surface homeomorphisms can assist in designing efficient “topologically chaotic” mixers. However, published results to date have been limited to 2D flows and quasi-3D protocols. Motivated by a simple stretching and folding argument used to derive stretching bounds in 2D flows (what Thurston describes as the iterate-and-guess method for constructing invariant train-tracks), we propose a topological approach to fully 3D fluid mixing. We consider periodic braiding of fluid in two orthogonal directions by inducing a flow with strategically placed ghost rods. The action of this braiding may be encoded by a transition matrix describing how certain area elements are mapped onto each other. The spectral radius of this matrix then furnishes an estimate of large-time asymptotic area growth rate. While this approach to mixing does not sit within the rigorous setting for TN theory, we find nonetheless that the predicted area stretch rates are very sharp for some model flows. Furthermore, we find that certain braids that are topologically trivial in 2D are quite effective in 3D.

Supported by Australian Research Council Discovery Grant DP0881054.

9:31AM GC.00008 Optimal Mixing, Part I, CHARLES R. DOERING, University of Michigan, ZHI LIN, Institute for Mathematics and its Applications, JEAN-LUC THIFFEAULT, University of Wisconsin — We investigate optimal incompressible stirring to mix an initially inhomogeneous distribution of diffusionless passive tracers. The H∞ Sobolev norm is adopted as the quantitative mixing measure of the tracer concentration field: its vanishing as t → ∞ is equivalent to the stirring flow’s mixing property in the sense of ergodic theory. We derive rigorous bounds on the rate of mixing by flows with fixed energy or energy dissipation rate constraints, and determine the flow field that instantaneously maximizes the decay of the mixing measure — when such a flow exists – by solving a variational problem. When no such ‘steepest descent’ flow exists (a possible but non-generic situation) we determine the flow that maximizes the growth rate of the scalar’s H∞ norm’s decay rate. This optimal stirring strategy is implemented numerically on a benchmark problem and compared to the rigorous bounds as well as an optimal control approach using a restricted set of flows.

Supported by NSF Awards PHY0555324, PHY0855335, and DMS-0806821.

9:44AM GC.00009 Optimal Mixing, Part II, ZHI LIN, Institute for Mathematics and its Applications, JEAN-LUC THIFFEAULT, University of Wisconsin, CHARLES DOERING, University of Michigan — We investigate optimal incompressible stirring to mix an initially inhomogeneous distribution of diffusionless passive tracers. The H∞ Sobolev norm is adopted as the quantitative mixing measure of the tracer concentration field: its vanishing as t → ∞ is equivalent to the stirring flow’s mixing property in the sense of ergodic theory. We derive rigorous bounds on the rate of mixing by flows with fixed energy or energy dissipation rate constraints, and determine the flow field that instantaneously maximizes the decay of the mixing measure — when such a flow exists – by solving a variational problem. When no such ‘steepest descent’ flow exists (a possible but non-generic situation) we determine the flow that maximizes the growth rate of the scalar’s H∞ norm’s decay rate. This optimal stirring strategy is implemented numerically on a benchmark problem and compared to the rigorous bounds as well as an optimal control approach using a restricted set of flows.

Supported by NSF Awards PHY0555324, PHY0855335, and DMS-0806821.

Monday, November 22, 2010 8:00AM - 10:10AM – Session GD CFD I: Immersed Boundary and Interface Methods Long Beach Convention Center 102B
UDAYKUMAR, University of Iowa Department of Mechanical and Industrial Engineering — One of the challenges involved with modeling organisms and other complex systems using CFD simulations lies in describing their complex geometries and motions with fidelity. We have developed a framework to overcome this difficulty by employing imagery as a basis from which to directly create such models. By combining nonlinear optical flow with image segmentation techniques, methodologies can also be applied to experimental flow velocimetry, and offer significantly enhanced spatial resolution compared with correlation methods used in particle image velocimetry. Preliminary application of the nonlinear optical flow methodology to planar fluid flow measurements will be demonstrated.

A numerical method is developed for simulating three-dimensional free surface flows in open channels of arbitrarily complex bathymetry. The complex geometry associated with that field. In the incompressible version of this method, the divergence-free condition of the velocity field is enforced throughout the fictitious domain by employing a two-phase flow level-set approach. A new method is developed for solving the level-set equations and the reinitialization equation in the context of the CURVIB framework. The method is validated for various free-surface model problems and its capabilities are demonstrated by applying to simulate turbulent free-surface flow in an open channel with embedded complex hydraulic structures.

CARLOS PANTANO, University of Illinois at Urbana-Champaign — We present a new immersed interface methodology for embedded complex geometries in Navier-Stokes flows over several simple, e.g., a sphere, and complex three-dimensional objects will be demonstrated.

RODOLFO ROSALES, Massachusetts Institute of Technology, JEAN-CHRISTOPHE NAVÉ, McGill University — We present a novel numerical method for solving the advection equation for a level set function. The new method uses Hierarchical-Gradient Truncation and Remapping (H-GTr) of the original PDE. Our strategy reduces the original PDE to a set of decoupled linear ODEs with constant coefficients. Additionally, we introduce a remapping strategy used to periodically guarantee solution accuracy. The resulting scheme is unconditionally stable, and the solution accuracy is nearly independent of the time step. We will evaluate our method in 2D and present results to several classical benchmark problems.

9:05AM GD.00006 The immersed interface method for 3D rigid objects in a flow1, SHENG XU, Southern Methodist University — In the immersed interface method, an object moving in a fluid is treated as the fluid enclosed by a singular force, and the singular force enters numerical schemes through jump conditions. In this talk, I will present a boundary condition capturing approach to determine the singular force for a 3D moving rigid object. Unlike many ad hoc penalty approaches, this approach is explicit but numerically stable. I will demonstrate its accuracy, stability and efficiency using flow due to an oscillating sphere/torus and flow due to a flapping wing.

1 This work is supported by NSF grant DMS 0915237.

9:18AM GD.00007 Optical Flow-Based Modeling and Velocimetry. SETH DILLARD, JAMES BUHHOLZ, H.S. UDAYKUMAR, University of Iowa Department of Mechanical and Industrial Engineering — One of the challenges involved with modeling organisms and other complex systems using CFD simulations lies in describing their complex geometries and motions with fidelity. We have developed a framework to overcome this difficulty by employing imagery as a basis from which to directly create such models. By combining nonlinear optical flow with image segmentation techniques, we are able to generate a level set field that moves under the influence of optical flow vectors computed on an image sequence, and thereby supply an immersed boundary to our flow solver. All of these operations take place on a fixed Cartesian mesh, obviating the complexities associated with fitted grid methods. These methodologies can also be applied to experimental flow velocimetry, and offer significantly enhanced spatial resolution compared with correlation methods used in particle image velocimetry. Preliminary application of the nonlinear optical flow methodology to planar fluid flow measurements will be demonstrated.

9:31AM GD.00008 The Immersed Interface Method for Two-Fluid Flows, MIGUEL UH, Southern Methodist University — In the Immersed Interface Method, a two-fluid flow problem is formulated as one set of governing equations and simulated on a fixed Cartesian grid. The effect of the two-fluid interface enters the formulation as a singular force and a numerical scheme as jump conditions. In this talk, we will present principal jump conditions and discuss the difficulties in implementing them. Finally, we will demonstrate the accuracy and efficiency of our immersed interface method for two-fluid flow simulation.

1 This work is supported by NSF grant DMS 0915237.
9:44AM GD.00009 A Direct-Forcing Immersed Boundary Method with Dynamic Velocity Interpolation, RANDALL MCDERMOTT, NIST — In a direct-forcing immersed boundary method, first introduced by Fadlun et al. (2000), the momentum equation is supplemented by a force term which drives the local velocity to a specified value. The method has gained popularity due to its ease of implementation in Cartesian, structured flow solvers and several variants on the basic theme have been proposed (see, e.g., Balaras (2004), Choi and Edwards (2008), Roman et al. (2009)). Generally, the first off-wall velocity point is forced to obey a simple interpolation, usually a linear, power-law, or log-law profile. These methods have been shown to work well for incompressible, statistically stationary flows. In the method proposed here, the velocity is obtained through dynamic evaluation of the boundary layer equations. The advantage of this approach is that, in principle, it is possible to better control the flow divergence (important for variable-density flows like fire—the primary application of our solver) and to account for the effects of local fluctuations in the flow field. The boundary layer equations are discretized with a second-order spatial scheme and time-step restrictions are avoided by formulating the streamwise momentum equation as an ordinary differential equation in time with a simple analytical solution. The method is tested on wavy-channel and cylinder/sphere wake flows across a broad range of Reynolds numbers.

9:57AM GD.00010 A new immersed interface method applied to hydrodynamics of a fusion chamber1, RICHARD KRAMER, CARLOS PANTANO, University of Illinois at Urbana-Champaign, GWEN LOOSMORE, ANDREW COOK, Lawrence Livermore National Laboratory — A new immersed interface technique is discussed for a high-order Cartesian fluid solver, with results presented from a range of problems. The new approach is designed to generate smooth fields in the ghost (fictitious) regions of the domain occupied by solid objects. A parallel implementation of this approach and its coupling to very complex geometries is discussed. We discuss some preliminary results for jet cooling inside a chamber containing Xenon at temperatures around 8000 K. The compressible, turbulent and radiative environment is modeled using the LLNL code Miranda with the new immersed interface technique used to represent the chamber walls.

Monday, November 22, 2010 8:00AM - 10:10AM – Session GE Instability: Jets and Wakes I Long Beach Convention Center 102C

8:00AM GE.00001 Bifurcation and stability analysis of a jet in crossflow at low velocity ratios, MILOŠ ILAK, PHILIPP SCHLATTER, SHERVIN BAGHERI, DAN HENNINGSON, Linne Flow Centre, Dept. of Mechanics, Royal Institute of Technology (KTH), SE-10044 Stockholm, Sweden — We study an incompressible jet in crossflow at low values of the jet-to-crossflow velocity ratio R using Direct Numerical Simulation and linear stability and sensitivity analysis. A Hopf bifurcation is found to occur slightly above R = 0.75, and the frequency of the limit cycle oscillation is the frequency of the unstable linear eigenmode. We find that the frequency of the limit cycle is the same as that of the most unstable eigenmode even for higher values of R, for which multiple unstable modes are found from the linear stability analysis. Using the leading adjoint global eigenmode near the first bifurcation point, we identify the location of the “wavemaker,” i.e., the region in the flow most sensitive to localized feedback. This region, interpreted as the core of the first instability, is found to be in the shear layer that surrounds the counter-rotating vortex pair which is the dominating feature of the dynamics of the jet. Furthermore, we perform a sensitivity analysis of the most unstable eigenvalue near the bifurcation point using the first direct-adjoint eigenmode pair.

8:13AM GE.00002 Unsteady Simulation of an ASME Venturi Flow in a Cross Flow, JEREMY BONIFACIO1, CEERS/CSULB, HAMID RAHAI2, MAE Dept., California State University, Long Beach — Unsteady numerical simulations of an ASME venturi flow into a cross flow were performed. The velocity ratios between the venturi flow and the free stream were 25, 50, and 75%. Two cases of the venturi with and without a tube extension have been investigated. The tube extension length was approximately 4D (here D is the inner diameter of the venturi’s outlet), connecting the venturi to the bottom surface of the numerical wind tunnel. A finite volume approach with the Wilcox K-ω turbulence model were used. Results that include contours of the mean velocity, velocity vector, turbulent kinetic energy, pressure and vortices within the venturi as well as downstream in the interaction region indicate that when the venturi is flushed with the surface, there is evidence of flow separation within the venturi, near the outlet. However, when the tube extension was added, the pressure recovery was sustained and flow separation within the venturi was not present and the characteristics of the flow in the interaction region were similar to the corresponding characteristics of a pipe jet in a cross flow.

8:26AM GE.00003 Low Density Transverse Jet Shear Layer Instabilities and their Control, DANIEL GETSINGER, CORY HENDRICKSON, AARON SUNG, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — This work describes an experimental characterization of the instabilities occurring in the near-field of the variable density transverse jet shear layer. Jets composed of mixtures of helium and nitrogen are injected from a converging nozzle mounted flush with an injection wall, issuing into an air crossflow. The jet-to-crossflow density ratio S is varied between 1.00 and 0.14 by changing the proportions of nitrogen and helium. Jet-to-crossflow momentum flux ratios J are varied in the range \( \infty > J > 2 \) at each value of S. The results of hotwire anemometry measurements in the jet shear layer indicate that a transition from convective to absolute instability has dependence on the jet-to-crossflow momentum flux ratio J and separate dependence upon the density ratio S. This transition, in a similar fashion to that examined in the equidensity transverse jet and in other types of shear flows, is characterized by several clear spectral features, including sharp spectral peaks, resistance to low level acoustic forcing for the globally unstable (self-excited) case, and broadband oscillations with high receptivity to applied forcing for the convectively unstable case. The presence of crossflow is observed to alter the global instability transition classically observed in low density free jets, allowing it to occur at higher density ratios S as J is reduced.

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

1Supported by NSF grant CBET-0755104 and the NASA GSRP

2Davitian, et al., JFM, to appear, 2010
Sensitivity analysis of low-density reacting jets, GARY CHANDLER, University of Bristol, JOSEPH NICHOLS, Center of Turbulence Research, Stanford University, MATTHEW JUNIPER, Department of Engineering, University of Cambridge, PETER SCHMID, Laboratoire d’Hydrodynamique (LaDHyX), CNRS-Ecole Polytechnique — A Low-Mach-number formulation of the Navier–Stokes equations is used to simulate an axisymmetric low-density jet diffusion flame that exits into stationary surroundings through a hole in a flat solid wall. A lifted flame that is marginally-stable in a hydrodynamic sense is considered. The equations are linearized about a steady solution of the nonlinear system and a corresponding set of adjoint equations is formed. Direct-linear and adjoint global modes are found with direct numerical simulation (DNS) and provide a map of the most sensitive locations to external forcing and external heating. Acoustic excitation is modeled as an external force in the momentum equation and a map of the most sensitive locations of the flame to acoustic excitation is given. The most sensitive locations to force feedback and to heat and drag from a hot-wire are then analyzed. Force feedback can occur from the placement of a sensor-actuator in the flow or can be considered as a mechanism for global instability. The lifted flame is particularly sensitive to outside disturbances and acoustic forcing in the non-reacting zone.

Global modes of compressible subsonic jets, XAVIER GARNAUD, LUTZ LESSHAFFT, PETER SCHMID, PATRICK HUERRE, LaDHyX, CNRS - Ecole Polytechnique — Global instability modes are computed for spatially developing jets at high subsonic Mach number. Both isothermal and hot configurations are considered. The jet exits a cylindrical nozzle, which is included in the numerical domain. Particular attention is directed to the aero-acoustic features of the jet, and the acoustic far-field is resolved as part of the global mode. Accurate resolution of sound propagation requires large computational domains, as well as high-order discretization schemes, which is numerically challenging with existing techniques, in particular in terms of memory requirements. We present a novel method for the computation of direct and adjoint eigenmodes of the global instability problem. Temporal filtering, applied to a time-stepping approach, allows to extract user-selected modes at significantly lower computational cost than common matrix-based techniques.

Analysis of time-resolved tomographic PIV data of a transitional jet, PETER SCHMID, LaDHyX, Ecole Polytechnique, France, DANIELE VIOLATO, FULVIO SCARANO, TU Delft, The Netherlands — Time-resolved tomographic particle image velocimetry (TR-TOMO-PIV) data of a transitional water jet at a Reynolds number of Re = 1500 have been obtained capturing all relevant spatial and temporal scales. These flow fields have then been processed by the dynamic mode decomposition (DMD) which extracts frequencies and associated coherent structures that constitute a significant part of the overall dynamics. Three data sets, covering the primary instability near the nozzle, the rise of secondary features further downstream and the breakdown into turbulence, have been analyzed, and frequency distributions and principal flow structures will be presented. Besides a temporal analysis of the data, a spatial analysis (in the axial direction) will be performed and compared to the findings from the temporal framework.

Swirling Turbulent Jet Structure Characterization Using Proper Orthogonal Decomposition of Flow Visualization Images, JONATHAN NAUGHTON, RICHARD SEMAAN, University of Wyoming — Planar Laser Scattering (PLS) was used to acquire instantaneous cross-sectional images of a swirling jet. The jet was seeded with small oil droplets whereas the ambient air was unseeded. Light from a laser sheet passing through the jet was scattered from the droplets and imaged using a camera. Proper Orthogonal Decomposition (POD) was applied to ~1000 images to determine the energy containing structure in the flow. A POD implementation was used that took advantage of axisymmetry to assure structure consistent with the flow. The results indicate that jets with a degree of swirl that exceeded a certain threshold exhibit an increased importance of the second azimuthal mode in the near-field as compared to a non-swirling jet. At distances further downstream, the mode two dominance decreases and mode one has an increased importance. Reconstructions of the swirling jet using the dominant modes provides evidence that the swirling jet contains very different turbulent structure in the near field as compared to non-swirling jets. The findings of this work are consistent with recent experimental and theoretical studies and provide guidance for future studies characterizing the structure responsible for swirling jet’s unique behavior.

Self excited oscillations in swirling jets: Stability analysis and empirical mode construction, KILIAN OBERTHEINER, MORITZ SIEBER, CHRISTIAN NAVID NAYERI, CHRISTIAN OLIVER PASCHERTEIT, Technical University Berlin, Germany, CHRISTOPH PETZ, HANS CHRISTIAN HEGE, Zuse Institut, Berlin, Germany, BERND NOACK, Institut Primo, Poitiers, France, ISRAEL WYGNANSKI, University of Arizona, Tucson, USA — Swirling jets undergoing vortex breakdown are known to be dominated by strong harmonic oscillations. Our experiments suggest the existence of a self-excited global mode having a single dominant frequency. The wave-maker of this oscillatory mode is found to be located in the jet center causing the swirling jet to precess. The oscillations trigger a convectively unstable co-rotating counter-winding helical structure that is located at the periphery of the recirculation zone. The resulting time-periodic 3D velocity field is predicted theoretically by employing linear stability analysis. It compares remarkably well to empirical 3D-modes that were constructed from uncorrelated 2D snapshots of PIV data, using proper orthogonal decomposition (POD). Stability analysis is further employed to detect regions of absolute instability and to derive the temporal growth-rate of the global mode. Results are compared to time-resolved measurements of the transient growth of the global mode.

Acoustic-gravity waves generated by wake flows, CHRISTOPE MILLET, CEA, DAM, DIF, F-91297 Arpajon, France, STEPHANE LE DIZÉS, IRPHE, CNRS, F-13384 Marseille cedex 13, France — The wavy wall analog framework is used to obtain a model problem for the acoustic-gravity wave field generated by a three-dimensional wave packet, that may be seen as a model for wake flow instabilities. In this stability, we develop a numerical method which is highly scalable, and more specifically, we estimate the propagation of acoustic and gravity waves in terms of saddle-point contributions. The saddle-points are computed from the general dispersion relation that we deduce from a compressible model with earth rotation and non-Boussinesq effects. Particular attention is paid to the far-field limit for which a single saddle-point contribution enables the description of both acoustic and gravity waves, also depending on the streamwise phase velocity of the wake packet. The transition from low-frequency acoustic waves (or infrasounds) to gravity waves can be treated in the same way as the acoustic radiation of supersonic two-dimensional shear layers.

Evolution of the turbulent/non-turbulent interface of an axisymmetric turbulent jet, M. KHASHEHCHI, A. OOI, I. MARUSIC, Department of Mechanical Engineering, University of Melbourne, Victoria, Australië, J. SORIA, Laboratoire for Turbulence Research in Aerospace and Combustion, Dept. of Mechanical and Aerospace Engineering, Monash University, Victoria, Australia — Measurements of a turbulent round air jet, using Particle Image Velocimetry (PIV), were made in order to investigate the dynamics and transport processes at the continuous and well-defined bounding interface between the turbulent and non-turbulent regions (T/NT) of the flow. The jet Reynolds number was $Re_D = 3000$ and the measurements were made between 0 and 50 nozzle diameters from the nozzle exit. A velocity thresholding technique was used and found to compare well against available results obtained using similar detection criterion reported in the literature. The evolution of the coherent turbulent structures at the jet exit as the jet expands through the nozzle, as indicated by the conditionally averaged streamwise velocity, azimuthal vorticity, turbulent intensity and Reynolds shear stress across the interface. A clear change in behaviour is noted going from the near-field region, $z/d = 0$ to 8, to the far field (self-similar) region. These will be described and discussed.
8:00AM GF.00001 Wake modes of rotationally oscillating cylinders, PRABU SELLAPPAN, TAIT POTTEBAUM, University of Southern California — Vortex shedding from bluff bodies is a fundamental problem in fluid mechanics and is important in applications such as vortex-induced vibration, heat transfer, and as a test for control strategies. Prior work has focused on vortex shedding from cylinders in cross flow and cylinders undergoing transverse or streamwise oscillations. Vortex shedding from a rotationally oscillating cylinder and the different wake modes that are produced have been investigated in this study. Experiments were carried out in a water tunnel at Re=750 for various amplitudes and frequencies of rotational oscillations. DPIV was used to study and map the different wake modes within the parameter space. Results show the mapping of wake modes to regions of the parameter space ranging from 0.7 to 3 times the natural shedding frequency and peak-to-peak rotational oscillation amplitudes from $5^\circ$ to $160^\circ$. The wake modes and the regions in which they occur are compared with scalar measurements previously reported in the literature. Further utilization of the wake mode map will include understanding heat transfer from rotationally oscillating cylinders.

8:13AM GF.00002 The relationship between Strouhal number and Reynolds number for a heated cylinder in the shear layer instability regime, JOSEPH LAURIENTI, TAIT POTTEBAUM, University of Southern California — The wake structure of a circular cylinder in isothermal cross flow has been extensively studied, and general agreement exists in the literature on the relationship between Strouhal number ($S_t$) and Reynolds number ($Re$) for parallel vortex shedding. However, no such consensus relationship exists for a heated cylinder in cross flow. Some recent studies have examined the $S_t$–$Re$ relationship for heated cylinders in the laminar vortex shedding regime and have successfully collapsed the data from various temperature ratios ($T^* = T_{cyl}/T_{inf}$) using an effective Reynolds number ($Re_{eff}$) that evaluates fluid viscosity at a temperature between the free stream and cylinder temperatures. The present work focuses on higher $Re$, where the separated shear layers become unstable. Water tunnel experiments were performed on parallel vortex shedding from a heated cylinder in the range $250 < Re < 800$ for various $T^*$. Long duration DPIV data sets were used to measure both vortex shedding frequency and detailed wake structure as functions of $Re$ and $T^*$. $S_t$–$Re$ curves will be presented for each $T^*$ and the use of $Re_{eff}$ to collapse the data will be evaluated.

8:26AM GF.00003 An experimental study of flow past two rotating cylinders, SANJAY KUMAR, BENITO GONZALEZ, Department of Engineering, The University of Texas at Brownsville, OLIVER PROBST, Physics Department, Technologico de Monterrey (Mexico) — Flow past two uniformly rotating cylinders in a side-by-side configuration is studied experimentally at Reynolds numbers, $Re$, varying from 100 to 500 and the ratio of surface speed of cylinder to the free stream velocity, $\alpha$, varying from 0 to 5. The center-to-center spacing between the cylinders, $T$, normalized by the cylinder diameter, $D$ are 1.8, 2.5, 4.0, and 7.5. Two possibilities of rotations are considered with the cylinder surfaces in between the two cylinders moving upstream in one case (inward rotation case) and downstream in the other (outward rotation case). The diagnostics is done by flow visualization and particle-image-velocimetry. Vortex shedding is found to be suppressed in the inward rotation cases for $Re = 200$ to 500 and all spacing ratios at $\alpha = \alpha_s \sim 2.0$. The value of $\alpha_s$, for $Re$ of 100 in this case increases from 1.2 to 1.7 as $T/D$ increases from 1.8 to 4.0 and does not increase further with $T/D$. For outward rotation cases, vortex shedding suppression is observed for $Re$ of 100 and for all values of $T/D$; however, for higher $Re$, suppression is observed for $T/D$ of 4.0 and 7.5 only. The measurements of $\alpha_s$ in this case showed a decreasing trend with increasing $T/D$. Symmetry breaking in the wake is reported for inward rotation case near $\alpha_s = 1.35$ for the case of $T/D = 2.5$ at $Re$ of 200.

8:39AM GF.00004 Secondary Instability in the Flow Past Two Aligned Square Cylinders1, CHOON-BUM CHOI, Inha University, Korea, YONG-JUN JANG, Korea Railroad Research Institute, Korea, KYUNG-BOO YANG, HYUNJUN JEON, Inha University, Korea — Interference of the wakes behind two nearby bluff bodies is important in many engineering applications. In this investigation, secondary instability (SI) in the flow past two square cylinders in side-by-side or tandem arrangements has been numerically studied via a Floquet analysis. An immersed boundary method was employed to implement the cylinders in the computational domain. The distance between the neighboring faces of the two cylinders (G) is the key parameter which affects SI under consideration. In this presentation, we report the critical Reynolds number for SI and the corresponding spanwise wave number of the most unstable (or least stable) wave for each of the selected Gs. Several distinct modes were identified in both arrangements, and described in detail. The representative three-dimensional vortical structure of each mode was depicted with vorticity contours. We also attempted to explain the underlying mechanisms of the key features of the secondary instability from the view points of flow physics. 1This work was supported by Korea Railroad Research Institute (Research Title: The study of standardization for urban railway facilities (SW10002)).

8:52AM GF.00005 Characteristics of the flow over a sphere at subcritical Reynolds numbers2, JUNGIL LEE, KWANGMIN SON, HAEcheon Choi, Seoul National University — The characteristics of turbulent flow over a sphere at subcritical Reynolds numbers is investigated using the mode analysis. The Reynolds numbers considered are $Re = 3700$, $10^4$ and $10^5$. The flow fields are generated from large eddy simulation with a dynamic global subgrid-scale model based on the Germano identity [Park et al., Phys. Fluids (2006); Lee et al., Phys. Fluids (2010)]. The flow statistics are in excellent agreement with previous experimental and numerical ones. The mode analysis is conducted on the axial velocity fluctuations integrated over the radial direction at each streamwise location. The axisymmetric mode (mode 0) represents cylindrical vortex rings or sheet that envelop(s) the recirculation region, and the helical mode (mode 1) is related to hairpin vortex or wavy vortical structure in the wake. The energy at each mode is maximum near the end of the recirculation region and decreases downstream. At $Re = 3700$, mode 0 is dominant within the recirculation region but mode 1 becomes dominant in downstream locations. On the other hand, at $Re = 10^4$ and $10^5$, mode 1 is most dominant throughout the flow field. These features are also manifest from instantaneous vortical structures.

1Supported by the NRL and WCU Programs, KRF, MEST, Korea.

9:05AM GF.00006 Turbulent Boundary Layer Separation Induced over a Flat Plate by a Rotating Cylinder1, FARHANA AFROZ, EMILY JONES, DREW SMITH, JENNIFER WHEELUS, AMY LANG, University of Alabama — A novel technique to generate and control an adverse pressure gradient (APG) over a flat plate was implemented by using a rotating cylinder for the purpose of studying turbulent boundary layer (TBL) separation. For this experiment, a flat plate and a fixed diameter cylinder were mounted vertically in a water tunnel to investigate the flow field where the boundary layer was tripped to the turbulent state. Variability in the strength of the APG induced on the plate was achieved using the rotation speed of the cylinder. Digital Particle Image Velocimetry (DPIV) was used to investigate the nature and extent of TBL separation induced by the cylinder rotation. Moreover, a theoretical, inviscid flow calculation of the pressure coefficient induced by the rotating cylinder on the flat plate was performed to predict the strength of the APG. Location of separation, percentage mass flow reversal, and length of the separated flow region were all analyzed as a function of the Reynolds number and strength of the APG.

1Funding under NSF CBET grant 0932352, NASA AL-EPSCoR and the Lindbergh Foundation is gratefully acknowledged.
9:18AM GF.00007 Flow Instability in Baffled Channel Flow1, CHANGWOO KANG, KYUNG-SOO YANG, KY-ONGJUN LEE, Inha University, Korea — Flow instability of baffled channel flow, where thin baffles are mounted on both channel walls periodically in the direction of the main flow, has been numerically investigated. Flow in a baffled channel is regarded as a simple model for flow in finned heat exchangers, including micro channels. In baffled channel flow, flow characteristics are significantly affected by geometrical configuration of the baffles. Two key parameters were considered, namely baffle interval (L) and Reynolds number (Re) of the main flow. The baffle height is fixed as one quarter of the channel height (H). By using a parametric study, we elucidate dependency of the primary instability, a Hopf bifurcation from steady to a time-periodic flow, on L. It turned out that the most unstable flow is obtained with L/H=3. Transition of two-dimensional (2D) time-periodic flow to three-dimensional (3D) flow is initiated by a secondary instability (SI). Floquet stability analysis was performed to identify the critical Reynolds number of SI for some selected baffle intervals. Several distinct modes were identified, and dependency of SI on L was elucidated. A 3D simulation was finally carried out to confirm the Floquet analysis. The current results shed light on understanding flow characteristics of a finned heat exchanger.

1This work was supported by UVRC, Korea.

9:31AM GF.00008 An Experimental Study of Flow Separation over a Flat Plate with 2D Transverse Grooves1, EMILY JONES, AMY LANG, FARHANA AFROZ, JENNIFER WHEELUS, DREW SMITH, University of Alabama — It has been hypothesized that flexible shark scales may aid in controlling boundary layer separation in that the scales bristle when encountering a localized flow reversal, thereby forming cavities within the skin that trap vortices between the scales. The formation of the embedded vortices can lead to the creation of a partial slip condition over the surface as well as turbulence augmentation in the boundary layer. In an attempt to replicate and study these effects on flow separation, a simplified model of the shark skin consisting of a plate with square 2D transverse grooves was utilized. Separation over the plate was induced via the placement of a rotating cylinder above the surface, and the experiments were carried out in a water tunnel with a tripped turbulent boundary layer. Using DPIV to analyze the flow, the results were compared to separation occurring over a flat plate. The effects on the location of separation and length of the separated flow region were all analyzed as a function of the Reynolds number and strength of the adverse pressure gradient induced by the rotating cylinder.

1Work performed under REU site sponsored by NSF grant EEC 0754117.

9:44AM GF.00009 Investigating Separated Shear Layer Development over an Airfoil with an Imbedded Microphone Array, SERHIY YARUSEVYCH, RYAN GERAKOPULOS, University of Waterloo — At low Reynolds numbers, laminar boundary layer separation on an airfoil often leads to deterioration in airfoil performance and noise emissions. The development of a separated shear layer is governed by laminar to turbulent transition, involving formation of coherent structures. This study highlights the design of a time-resolved surface pressure measurement system capable of estimating salient flow characteristics based on the analysis of surface pressure fluctuations. Wind tunnel experiments were performed for a symmetric NACA 0018 aluminum airfoil model equipped with a total of 95 static pressure taps and 24 microphones. Tests were performed for a range of angles of attack and Reynolds numbers to investigate two flow regimes common to airfoils operating at low Reynolds numbers, namely, flow separation without subsequent reattachment and separation bubble. Experimental results show that the microphones can be utilized to estimate the extent of the separation region and study the development of flow disturbances in the separated shear layer. Using hot wire measurements for validation, it is demonstrated that the microphones can detect the frequency signature of disturbances amplified in the separated shear layer. Further statistical analysis is employed to estimate such important characteristics of the disturbances and coherent structures as spanwise correlation, propagation speed, and phase.

9:57AM GF.00010 Experimental study of unsteady turbulent boundary layer separation under conditions relevant to dynamic stall1, DAVID SCHATZMAN, FLINT THOMAS, University of Notre Dame — An experimental investigation focused on the study of the physics of unsteady turbulent boundary layer separation under conditions relevant to the dynamic stall phenomenon is presented. A flat boundary layer development plate allows for the growth of a turbulent boundary layer of thickness sufficient for high spatial resolution measurements. Downstream of the flat plate, a convex ramp section imposes a streamwise adverse pressure gradient that gives rise to boundary layer separation. In order to impose an unsteady pressure gradient, an airfoil section is located above the convex ramp. Leading edge plasma flow control is used to alternately attach and separate the airfoil flow which gives rise to unsteady turbulent boundary layer separation on the convex ramp. Measurements of the resulting unsteady turbulent boundary layer separation via phase-locked two-component PIV, unsteady surface pressure measurements, and wall-mounted hot-films quantify the dynamics of the separation process at the wall and throughout the unsteady boundary layer. Two-component LDA measurements are used to characterize the motions of ejection and sweep events within the unsteady boundary layer using a quadrant splitting technique.

1Supported by ARO W911NF-07-0122

Monday, November 22, 2010 8:00AM - 10:10AM –
Session GG GFD: Atmospheric Flows I Long Beach Convention Center 103B

8:00AM GG.00001 ABSTRACT WITHDRAWN –

8:13AM GG.00002 A dynamic roughness model for LES of flow over multiscale, fractal-like surfaces: application to synthetic and real topography1, WILLIAM ANDERSON, CHARLES MENNEVEAU, Johns Hopkins University, Baltimore, MD — The topography of many natural surfaces encountered in geophysical flows is known to be multiscale and fractal-like. We present the so-called dynamic surface roughness (DSR) model, a framework for representation of drag effects imposed on a high- Reynolds number boundary layer by a multiscale rough surface. In developing and testing the DSR model, we consider synthetic fine-grained multiscale surfaces with spectral exponent βs ranging between -3.0 (smoothest) to -1.2 (roughest). The fine-grained surface is spatially filtered at the large-eddy simulation (LES) resolution, resulting in a resolvable and subgrid-scale (SGS) component. The SGS component is represented with an effective roughness length. The effective roughness length is written as the product of local SGS roughness root-mean-square and an unknown roughness index α. We dynamically evaluate α with a self-consistency condition applied to the plane-average of total wall stress resolved at the grid- and a test-filter width. Results for flow over synthetic surfaces indicate strong dependence of α on surface spectral exponent. We also apply the DSR model to LES of flow over a real landscape using digital elevation map data from the USGS.

1Supported by NSF (EAR-0609090).
8:26AM GG.00003 Applying renormalized numerical simulation to model turbulent flow over a fractal tree canopy. JASON GRAHAM, CHARLES MENEVEAU, Johns Hopkins University, Baltimore, MD — Renormalized Numerical Simulation (RNS) is a down-scaling approach that uses drag forces from resolved flow fields to parameterize the drag forces due to unresolved scales (Chester et al., 2007, J. Comp. Phys.). The RNS procedure is analogous to the dynamic sub-grid scale model. In RNS a form drag model is used to parameterize the forces and the drag coefficient, $c_D$, is dynamically evaluated by learning from the large scale problem and recursively feeding back to the small scale problem the renormalized drag forces. In this study a suite of Large Eddy Simulations using RNS are performed to analyze boundary layer flow over a canopy of fractal trees. The fractal trees provide complex boundary-turbulence interactions while maintaining tractable characteristics that can be systematically studied. Resolved branches are represented in the LES using the immersed boundary method. Several RNS implementations are tested and compared: 1) explicit and 2) implicit time formulations, and two spatial treatments for $c_D$: 1) local 2) global definitions. For these set of simulations the time averaged flow field, Reynolds and dispersive stresses, and drag forces of the canopy are computed.

8:39AM GG.00004 Challenges in large-eddy simulation of cumulus convection. GEORGIOS MATHEOU, DANIEL CHUNG, Jet Propulsion Laboratory/California Institute of Technology, LOUISE NUIJENS, BJORN STEVENS, Max Planck Institute for Meteorology, JOAO TEIXEIRA, Jet Propulsion Laboratory/California Institute of Technology — High-resolution simulation is a vital tool for studying the physical processes in the atmospheric boundary layer. In spite of the numerous encouraging large-eddy simulation (LES) results, prediction of complex turbulent flows continues to present many challenges. The present study considers the impact of various choices pertaining to the numerical solution of the governing equations on the LES prediction and the association of these choices to flow physics. Simulations corresponding to the trade wind precipitating shallow cumulus composite case of the Rain In Cumulus over the Ocean (RICO) field experiment were carried out. Global boundary layer quantities such as cloud cover, surface precipitation rate, power spectra and the overall convection structure were used to compare the effects of different discretization implementations, grid resolution and computational-domain size. The different discretization implementations were found to exert a significant impact on the LES prediction. The observed differences can be attributed to the nonlinear nature of moist convection, especially when precipitation is present, which results in an increased sensitivity of the atmospheric boundary layer statistics to the representation of small-scale turbulence.

8:52AM GG.00005 Large-eddy Simulation of Turbulent Flows in an Urban Street Canyon. JEONG-MIN HWANG, BYUNG-GU KIM, CHANGHOON LEE, Yonsei University — Turbulent flow inside an urban street canyon is studied by means of large-eddy simulation. The simulated site is the ‘Teheran Street’ in Gangnam district of Seoul in Korea, which is one of the representative street canyon in Korea. The Reynolds number, based on the height of the tallest building in the domain and mean velocity there, is around ten million. The domain size is 600m in each direction, and tested grid size varies from 2m to 12m while typical small buildings are of order of 20 m. A constant Smagorinsky coefficient subgrid-scale (SGS) model is used. Performance of the SGS model for various resolutions is assessed by investigating shallow cumulus composite case of the Rain In Cumulus over the Ocean (RICO) field experiment were carried out. Global boundary layer quantities such as cloud cover, surface precipitation rate, power spectra and the overall convection structure were used to compare the effects of different discretization implementations, grid resolution and computational-domain size. The different discretization implementations were found to exert a significant impact on the LES prediction. The observed differences can be attributed to the nonlinear nature of moist convection, especially when precipitation is present, which results in an increased sensitivity of the atmospheric boundary layer statistics to the representation of small-scale turbulence.

9:05AM GG.00006 Large-eddy Simulation of the stable atmospheric boundary layer with explicit filtering and reconstruction. FOTINI CHOW, BOWEN ZHOU, University of California, Berkeley — Large-eddy simulation (LES) of the stably stratified atmospheric boundary layer is performed using an explicit filtering and reconstruction approach with a finite difference method. The dynamic reconstruction model (DRM) is used to represent the resolvable subfilter-scale and subgrid-scale stresses which make up the total turbulent stress. Several surface cooling rates are used, ranging from mildly stable to strongly stable intermittent turbulence cases. Two level jet develops with associated turbulent kinetic energy (TKE) generated around the top of the boundary layer, in agreement with field observations. The role of filtering on generation of this elevated TKE is explored. The ability of the DRM to represent energy backscatter from small to large scales is examined as a function of surface cooling and grid resolution. A turbulent bursting event is analyzed during intermittent conditions.

9:18AM GG.00007 Sensitivity Study of Contrail Development: Large Eddy Simulation and Parameterized Model. ALEXANDER NAIMAN, SANJIVA LELE, MARK JACOBSON, Stanford University — The development of aircraft condensation trails is sensitive to factors including ambient relative humidity, aircraft type, and environmental turbulence. The effect of these parameters on the transition from linear contrails to induced-cirrus clouds is a key uncertainty in estimating the impact of contrails on climate. A sensitivity study of these parameters has been conducted using a three-dimensional Large Eddy Simulation (LES) of the first twenty minutes of contrail development. The LES solves the incompressible Navier-Stokes equations with a Boussinesq approximation. The numerical scheme uses a second-order finite volume spatial discretization and an implicit fractional-step method for time advancement. Lagrangian contrail particles grow according to a model of ice deposition and sublimation. We present results in which turbulence, wind shear, and aircraft type were varied. Additional cases include variations in microphysical processes and in initial conditions. Results from the LES are compared to a simple parameterization of plume dynamics developed to model aircraft emissions in a global climate model. The parameterized model is shown to be valid for the late stages of the LES model results. Additional LES work will be required to validate the parameterized model to time horizons later than twenty minutes, which are relevant for the transition from linear contrail to induced cirrus.

9:31AM GG.00008 Modeling the Influence of Wind Characteristics and the Atmospheric Stability on Wind Turbine Performances. DARRON KORACIN, DRI; RADIAN BELU, Drexel University — The uncertainty of wind turbine performance measurements is closely related to the uncertainty of the wind velocity and other meteorological parameters. An inherent uncertainty in the power curve estimate is by using the wind speed measured at the hub height, as such considerable deviations often occur between the expected and produced power. Wind shear, direction changes, turbulence and atmospheric stability vary with height because of either meteorological and/or terrain conditions. The rotor size combined with the hub height of large turbines implies that turbines are often exposed to highly varying wind conditions (large wind and direction shears, turbulence and atmospheric stability) within the rotor span. These parameters will affect turbine structural safety and production. Velocity, temperature, and turbulence intensity data is generated using a model developed from Monin-Obukhov similarity theory and the $k$-$c$ turbulence model to resolve the atmospheric parameters (friction velocity, Monin-Obukhov length, temperature scale, and roughness length). The resulting nonlinear equations were solved numerically and tested against the observations. The rotor averaged wind speed is evaluated by numerically integrating the resulting velocity profile over the rotor area. Power output estimates were compared with the available data (manufacturers and literature) and are used in the turbine design.
9:44AM GH.00009 Visualizing the effects due to a wind turbine in a stratified turbulent boundary layer, NICHOLAS HAMILTON, RAUL BAYOAN CAL. Portland State University — As sustainable technologies and energy generation become more prolific, the need for larger wind farms becomes highly evident. It has been hypothesized that the behavior of heat and moisture transfer between the air and the ground is altered in the wakes of wind turbines. An experimental study at the Complex Boundary-layer and Wind Energy Based (CoWWEB) wind tunnel in Portland State University is performed to visualize the effects of these rotating structures under stratified conditions, thus modeling environments observed by a wind turbine array. A Schlieren technique is applied to study the interaction between the turbulent thermal boundary layer and the wind turbine. The Schlieren system employed here captures the temperature differences between the heat supplied through the floor of the wind tunnel and the air stream to image focal planes in upstream and downstream positions of a wind turbine. The data collected from this study demonstrates observable differences and effects due to the presence of the wind turbine.

9:57AM GH.00010 Insight on Turbulence Characteristics of an Urban-type Boundary Layer, BRUNO MONNIER, JONATHAN SWANSON, CANDACE WARK, Illinois Institute of Technology — An experimental investigation of the flow through an urban-type boundary layer (4 rows of 3 cuboid Plexiglas blocks) in an experimentally modeled atmospheric boundary layer will be presented. Stereoscopic PIV is utilized to obtain 3D flow characteristics of the flow within this complex geometry. The streamwise spacing of the array is chosen so as to mimic a common flow regime in urban areas, i.e. skimming flow regime. A large number of vertical planes distributed across the streets allows for a very good spatial description of the flow field at each Ra. Coherent structure identification tools are used to highlight the 3D patterns within each of the streets. A large number of PIV realizations in the domain provides valuable information about the flow field turbulence statistics as the flow is evolving from one street to the next. The incidence angle of the incoming flow field is also varied to assess the effect of flow channeling within the urban environment. Finally, two mean free stream speeds are studied to investigate the effect of the incoming wind profiles on the flow field turbulence.

1Supported by the Illinois NASA Space Grant Consortium.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GH Convection and Buoyancy Driven Flows IV Long Beach Convention Center 103C

8:00AM GH.00001 Homogeneous purely buoyancy driven turbulent flow, JAYWANT ARAKERI, Indian Institute of Science, Bangalore-12, MURALI CHOLEMARI, Indian Institute of Technology, Delhi, SHASHIKANT PAWAR, Indian Institute of Science, Bangalore-12 — An unstable density difference across a long vertical tube open at both ends leads to convection that is axially homogeneous with a linear density gradient. We report results from such tube convection experiments, with driving density caused by salt concentration difference or temperature difference. At high enough Rayleigh numbers (Ra) the convection is turbulent with zero mean flow and zero mean Reynolds shear stresses; thus turbulent production is purely by buoyancy. We observe different regimes of turbulent convection. At very high Ra the Nusselt number scales as the square root of the Rayleigh number, giving the so-called “ultimate regime” of convection predicted for Rayleigh-Bénard convection in limit of infinite Ra. Turbulent convection at intermediate Ra, the Nusselt number scales as Ra0.3. In both regimes, the flux and the Taylor scale Reynolds number are more than order of magnitude larger than those obtained in Rayleigh-Bénard convection. Absence of a mean flow makes this an ideal flow to study shear free turbulence near a wall.

8:13AM GH.00002 Viscous boundary layers in high Rayleigh number convection: A new insight from 3d velocity measurements, RONALD DU PUITS, Ilmenau University of Technology, LING LI, ANDRÉ THESS — The local transport inside the boundary layers in turbulent convection is one of the keys to understand the scaling of the global heat transport with respect to the temperature gradient and the vertical extent of a wall bounded fluid-mechanical system. We report highly resolved 3D-Laser Doppler Velocimetry measurements in a large-scale Rayleigh-Bénard experiment with air at Rayleigh numbers up to 10^12. The measurements were undertaken in the vicinity of the cooling plate in the central axis of the cylindrical sample. They differ from those reported in the paper du Puits et al [Phys. Rev. E 80, 036318 (2009)] in that all three velocity components have been measured simultaneously. In the present communication we will discuss the results of these measurements and compare them with previous ones as well as with theoretical predictions about the mean velocity profile and the fluctuations in non-isothermal shear layers.

8:26AM GH.00003 Dependence of the Nusselt number on the Rayleigh number for Prandtl numbers near 0.7, JAMES HOGG, GUENTER AHLERS, Department of Physics, University of California, Santa Barbara — We report Nusselt number measurements for a cylindrical Rayleigh-Bénard sample of height L = 96 cm and aspect ratio 1 = 0.079 that were made using three pure gases: helium (Prandtl number Pr=0.67), nitrogen (Pr=0.73), and argon (Pr=0.67-0.70) at pressures up to 47 bars. They cover the Rayleigh number range

0 \leq \gamma \leq 32

and the results for different gases disagree for relatively large  \gamma

and disagree for relatively small  \gamma

The uncorrected results are not well fit by the standard power law

\nu \propto \gamma^{0.7}

and the results for different gases disagree more than can be attributed to any expected Prandtl-number dependence. We find that a correction to the Nusselt number using a model for the non-linear phenomena. Average Nusselt and Sherwood numbers were evaluated while the convection patterns arise within the cavity. The wavy-wall was found to promote thermal stratification and low velocity multiple cell patterns for low buoyancy ratio.

1Supported by NSF Grant DMR07-02111.

8:39AM GH.00004 Viscous and Thermal Boundary Layers in Simulated Turbulent Rayleigh-Bénard Convection, JANET SCHEEL, ELISSA KIM, Occidental College — We present the results from numerical simulations of three-dimensional, fully turbulent Rayleigh-Bénard convection for cylindrical cells of aspect ratio 1 (diameter = depth). We use experimentally realistic boundary conditions, Prandtl numbers of 0.4 and 0.7, and Rayleigh numbers between 10^7 and 10^11. We focus on the thermal and viscous boundary layers, and compute profiles and boundary layer thicknesses in a variety of ways. We find that the different methods can effect the results. We also compare our results to experiments and theory.

8:52AM GH.00005 Transient buoyancy-driven flow in a vertical cylindrical enclosure with wavy sidewall due to thermal and concentration gradients, FAUSTO SANCHEZ, SIMON MARTINEZ, HUGO RAMIREZ, Universidad Autonoma de Nuevo Leon, ABRAMAM NAVARRO, Instituto Politecnico Nacional — An asymmetric transient convection flow, due to thermal and concentration gradients within a vertical cylindrical enclosure with adiabatic wavy sidewall, was studied. The two important cases of enclosure heated from below and heated from the top were studied. An analytical coordinate transformation was used to change the computation domain into a square. The heat and mass transfer were analyzed using non-dimensional parameters which include the cavity aspect ratio, the dimensionless wavelength and amplitude of the wavy-wall, Rayleigh and Prandtl numbers and the buoyancy ratio. For all cases the upper surface is consider as the one with high concentration, while the

1Authors thank CONACyT Projects 62054 and 103334, PAICyT-UANL and Redes 2008 PROMEP “Fuentes Renovables y Uso Eficiente de la Energía.”
9:05AM GH.00006 Natural convection in a cylindrical cavity. JOSE NUNEZ, MIGUEL LOPEZ, EDUARDO RAMOS, GUILLERMO HERNANDEZ, SERGIO CUEVAS. Center for Energy Research UNAM, MINERVÁ VARGAS, Instituto Tecnológico de Zacatepec. — Natural convection in a vertical cylinder heated from below is studied experimentally and numerically. The aspect ratio (diameter/height) is 1.3 and we observe convective motions for a Prandtl number of 6.66 and a range of Rayleigh numbers from $1.0 \times 10^5$ to $5.0 \times 10^5$. This range of Rayleigh numbers includes steady and time-dependent flows. Experimental observations were made with a coupled PIV system capable of simultaneously obtaining velocity distributions in two mutually perpendicular planes. The numerical model comprises the solution of the three-dimensional time-dependent Boussinesq equations in cylindrical coordinates. In all cases analyzed, the flows present complex three-dimensional structures and we use the vortex core concept as a visualization technique to characterize the fluid motion. Experimental observations are compared with theoretical calculations and quantitative agreement is obtained for steady flow and averaged values in unsteady flow.

9:18AM GH.00007 Axially periodic Rayleigh-Bénard convection in a cylindrical cell. LAURA SCHMIDT, University of Twente, The Netherlands, FEDERICO TOSCHI, Technische Universiteit Eindhoven, The Netherlands, ROBERTO VERZICCO, University of Rome - Tor Vergata, Italy, DETLEF LOHSE, University of Twente, The Netherlands. — Numerical simulations of Rayleigh-Bénard convection in an infinite cylindrical cell show that despite the restriction of velocity and temperature fluctuations due to the side walls, the system approaches the ultimate regime of thermal convection as the Rayleigh number (Ra) is increased. Here, Ra is defined based on the underlying linear temperature gradient which is driving the convection. This periodic system has exact solutions composed of modes of exponentially growing vertical velocity and temperature fields. In the low Ra regime these solutions dominate the dynamics and lead to very high and unsteady heat transfer. As Ra is increased, interaction between these modes stabilizes the system, evidenced by the increasing homogeneity and reduced fluctuations in the r.m.s. velocity and temperature fields.

9:31AM GH.00008 The boundary layer structure in Rayleigh-Bénard convection in a cylindrical cell. NANNI SHI, JOERG SCHUMACHER, TU Ilmenau, Germany. — We report first results of our studies of the boundary layer structure in turbulent Rayleigh-Bénard convection in a cylindrical cell of aspect ratio one. They are based on three-dimensional direct numerical simulations (DNS) of the Boussinesq equations at $Ra = 3 \times 10^6$ and $Pr = 0.7$. The study is motivated by two recent experiments: LDA measurements of the velocity boundary layer structure in the cylindrical barrel of Ilmenau by du Puits et al. and PIV measurements in a slender rectangular convection cell by Xia et al. Both experiments detected deviations from the classical Blasius solution for time-averaged flow profiles. A rescaling by the instantaneous boundary layer thickness resulted however in a much better agreement with the Blasius profile in case of the rectangular cell. The DNS allow us to combine the analysis methods of both experiments. We confirm the significant deviation for the time-averaged profiles. Closer agreement with the Blasius solution is also reproduced for the fit with the instantaneous thickness. Our analysis is extended to the Polihhausen solution in case of thermal boundary layer. The flow profiles are also taken at different positions in the boundary layers. Further statistical properties in both boundary layers are reported.

9:44AM GH.00009 3D pattern flow in a right-angled triangular cavity. RAFAEL CHAVEZ, FRANCISCO J. SOLORIO, Department of Thermofluids, Engineering School, UNAM. — Most numerical studies in triangular cavities had been carried out considering the flow as two-dimensional. In the last years some numerical studies have been made to take in account the three-dimensional behavior, but there is a lack in experimental work in the field of right-angled triangular cavities. This work is an effort to fill this lack. Particle image velocimetry (PIV) is used to study the flow pattern into a cavity with the inclined wall cooled, the vertical wall adiabatic and the horizontal bottom wall heated. Four Rayleigh numbers are considered: $5 \times 10^4$, $1 \times 10^5$, $5 \times 10^5$ and $1 \times 10^6$, and glycerin is used as working fluid. For the smallest Rayleigh number ($5 \times 10^4$) the flow is two-dimensional. As the Rayleigh number is increased, the flow evolves into a more complex three-dimensional pattern, with an array of cells whose rotation axes are normal to the vertical adiabatic wall. It is found that the number of cells depends on the Rayleigh number.

9:57AM GH.00010 Numerical simulation of the convective flow patterns within a rotating concentric annulus with radial gravity. ARES CABELLO, Facultad de Ingenieria, UNAM, RUBEN AVILA, Center for Aerospace Research and Education, UCI. — The GEODYNAMO research requires the numerical study of the natural convection of the fluid confined in a rotating spherical shell. We present the flow patterns of a uniform-density Boussinesq fluid within a rotating spherical annulus with radial aspect ratio $\eta = 0.35$. The convective flow is induced by a gravity field acting radially inward toward the center of the spheres, and the temperature difference between the internal sphere at $T_i$ and the external sphere at $T_e$ (where $T_i > T_e$). We also show (i) the influence of the rotation on the heat transfer rate, and (ii) the influence of the differential rotation (the internal sphere rotates at a different angular velocity than the reference frame and the external sphere) on the heat transfer rate. The fluid equations are solved by using the spectral element method (SEM). In order to avoid the singularity at the poles of the spheres, the numerical mesh is generated by using the Cubed-Sphere algorithm. The flow patterns are obtained for subcritical and supercritical Rayleigh numbers and Taylor numbers in the range $10^3$ and $10^5$. The results are successfully compared with data previously reported in the literature.

Monday, November 22, 2010 8:00AM - 9:57AM — Session GJ Flow Control IV Long Beach Convention Center 201A

8:00AM GJ.00001 Flow Structures and Interactions of a Fail-Safe Actuator. WASIF KHAN, YOSEPH ELIMELECH, MICHAEL AMITAY, Rensselaer Polytechnic Institute. — Vortex generators are passive devices that are commonly used in many aerodynamic applications. In their basic concept, they enhance mixing, reduce or mitigate flow separation; however, they cause drag penalties at off design conditions. Micro vanes implement the same basic idea of vortex generators but their physical dimensions are much smaller. To achieve the same effect on the baseline flow field, micro vanes are combined with an active flow control device, so their net effect is comparable to that of vortex generators when the active device is energized. As a result of their small size, micro vanes have significantly less drag penalty at off design conditions. This concept of dual-action is the reason why such actuation is commonly called hybrid or fail-safe actuation. The present study explores experimentally the flow interaction of a synthetic-jet with a micro vane in a zero pressure gradient flow over a flat plate. Using the stereo particle image velocimetry technique a parametric study was conducted, where the effects of the micro vane shape, height and its angle with respect to the flow were examined, at several blowing ratios and synthetic-jet configurations.
8:13AM  GJ.00002  Bluff Body Separation Control using Pulsed Actuation, GEORGE T.K. WOO, Georgia Institute of Technology, THOMAS M. CRITTENDEN, Virtual AeroSurface Technologies, Inc., ARI GLEZER, Georgia Institute of Technology — The severing of a separating shear layer using pulsed actuation jets is exploited for separation control over a 3D bluff body by the transitory manipulation of the shedding of the large-scale vortical structures. In the present wind tunnel investigation, actuation is effected by surface-integrated discrete arrays of pulsed (combustion-powered) actuators having a characteristic time scale that is an order of magnitude shorter than the convective time scale of the base flow. High-resolution PIV measurements, taken phase-locked to the actuation, show that the interaction between the pulsed jets and the separated cross flow results in significant streamwise generation of the vorticity downstream. The fluctuating shear layer then rolls up the induced fluid flow, and the resulting vortex sheet relaxes which directly affects the attachment process. Successive actuation can extend the interactions within the attaching boundary layer such that the aerodynamic forces and moments are quasi-steady and significantly enhanced compared to what can be achieved with continuous time-harmonic actuation. Supported by the US Army.

8:26AM  GJ.00003  Interaction of a Finite-span Synthetic-jet and Cross-flow over a Swept Wing, JOSEPH VASILE, YOSEPH ELIMELECH, MICHAEL AMITAY, Rensselaer Polytechnic Institute — An experimental investigation was performed to study the interaction of a single finite-span synthetic jet with the flow over a finite and swept back wing at a Reynolds number of 10^7 for three angles-of-attack. For the actuation levels, two momentum coefficients were considered, corresponding to two blowing ratios, 0.8 and 1.2. Stereoscopic PIV data were acquired in the vicinity of the synthetic-jet orifice at the wing’s mid-span section. The effect of blowing ratio was analyzed using both time and phase averaged statistics. The results show that the flow field in the vicinity of the synthetic-jet orifice becomes three-dimensional and time-dependent and is governed by the superposition of two kinds of flow structures: (1) streamwise structures that are associated with the finite span of the jet (edge vortices), and (2) spanwise flow structures that are generated along the orifice’s long axis due to the vortex pairs that are formed by the synthetic jet. Furthermore, an analysis of the flow field showed that the streamwise flow structures are more pronounced while the coherence of the spanwise flow structures is deteriorated within few orifice widths.

1Supported by AFOSR

8:39AM  GJ.00004  Hybrid Manipulation of Streamwise Vorticity in a Diffuser Boundary Layer, ABRAHAM GISSEN, BOJAN VUKASINOVIC, JOHN CULP, ARI GLEZER, Georgia Institute of Technology — The formation of streamwise vorticity concentrations by exploiting the interaction of surface-mounted passive (micro-vanes) and active (synthetic jets) flow control elements with the cross flow is investigated experimentally in a small-scale serpentine duct at high subsonic speeds (up to M = 0.6). Streamwise vortices can be a key element in the mitigation of the adverse effects on pressure recovery and distortion caused by the naturally occurring secondary flows in embedded propulsion systems with complex inlet geometries. Counter rotating and single-sense vortices are formed using conventional passive micro-vanes and active high-power synthetic jet actuators. Interaction of the flow control elements is examined through a hybrid actuation scheme whereby synthetic jet actuation augments the primary vanes’ vortices resulting in dynamic enhancement of their strength. It is shown that such sub-boundary layer individual vortices can merge and evolve into duct-scale vortical structures that counteract the inherent secondary flow and mitigates global flow distortion.

1Supported by NASA and the Boeing Company.

8:52AM  GJ.00005  Experimental study of airfoil separation control using synthetic jets, XI XIA, KAMRAN MOHSEN, CU Boulder — The flow control over an airfoil is studied experimentally in a wind tunnel. Synthetic jets are placed on the top surface of the airfoil as flow actuators. The position and the angle of the jet orifice, together with the frequency and jet strength could be varied in order to adjust the separation or reattachment points on the surface. An Array of hot-film sensors are located on the surface in order to detect the location of the reattachment point. The airfoil is mounted on a 6 d.o.f force balance system to dynamically measure the drag and lift forces on the airfoil. Results from the hot-film sensor array measurement are correlated to the measured drag and lift forces.

1Supported by NSF.

9:05AM  GJ.00006  Performance Enhancement of a Wind Turbine Blade using Synthetic Jets, KEITH TAYLOR, CHIA LEONG, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Recent developments in flow control techniques, coupled with increased interest in green energy technologies, have led to interest in applying flow control technologies to wind turbines, in an effort to increase power output and reduce structural stress associated with widely varying loading. A reduction in structural stress could lead to reduced operational costs associated with maintenance. Presented is an investigation into the effect of active flow control on the aerodynamic and structural aspects of a finite-span S-809 airfoil. Synthetic jets are employed in an open loop control scheme to demonstrate the effect on lift, drag, and vibrations of the blade at Reynolds numbers of 110,000 and 220,000. Vibrometer measurements are presented to quantify vibration frequency, and time dependent fluctuations in lift and drag are correlated to tip deflection fluctuations. Static and dynamic pitch conditions are examined, with a sinusoidal pitch profile implemented for dynamic conditions. It is demonstrated that flow control can reduce tip deflection in static conditions, as well as reduce or eliminate hysteresis as the blade dynamically pitches into and out of separation.

1Supported by AFOSR.

9:18AM  GJ.00007  Controlled Dynamic Stall using Pulsed Fluidic Actuation, GEORGE T.K. WOO, ARI GLEZER, Georgia Institute of Technology — Controlled attachment of transitory stall over an oscillating airfoil is investigated in wind tunnel experiments using a spanwise array of surface-integrated pulsed jet actuators such that the characteristic actuation time scale is an order of magnitude shorter than the convective time scale of the base flow. Earlier work showed that single-pulse actuation results in a rapid [O(10TCONV)] attachment of the separated flow followed by a slower [O(10TCONV)] detachment. These dynamics are exploited for controlled mitigation of pre- and post-stall dynamics during the oscillation cycle and the interaction between the actuation jets and the evolution of the dynamic stall vortex. The transitory effects of the actuation can be extended and exploited for trapping and regeneration of the dynamic stall vortex. The dynamic stall vortex is generated by using staged, multiple actuation pulses during the cycle. These interactions are investigated using high-resolution phase-locked PIV measurements in the cross stream plane (including the near wake). The temporal changes in the vorticity flux results in significant changes in circulation, and consequently in the time-dependent aerodynamic forces and moments.

3Supported by NASA.

9:31AM  GJ.00008  Stall Control Simulation with an Impulse Jet, SOL KEUN JEE, ROBERT MOSE, University of Texas at Austin, OMAR LOPEZ, Universidad de los Andes, Colombia — An impulse jet is investigated numerically to understand the mechanism by which this jet controls a stalled flow over an airfoil. The DDES (delayed detached eddy simulation) turbulence model is used in this stall control study for a NACA 4415 airfoil at an angle of attack of 20 degree and Reynolds number Re=570,000. An impulse jet, which is applied upstream of the nominal flow separation point, generates vortices that convect downstream, interact with the separating shear layer, dismantle the layer and allow following vortices to propagate along the surface in the separation region. These following vortices shift the separation point aft reattaching the boundary layer, which returns slowly to its initial stall condition, as observed in wind-tunnel experiments. A simple model of the impulse jet actuator used herein is found to be sufficient to represent the global effects of the jet on the stalled flow because it correctly represents the momentum injected into the flow.

3This work is supported by AFOSR grant FA9550-05-1-0411.
Numerical investigation of the interaction between a finite-span synthetic jet and a cross flow over a swept wing \(^1\), MICHEL RASQUIN, NICHOLAS MATI, ECAE, University of Colorado at Boulder, ONKAR SAINI, PECOS/ICES. The University of Texas at Austin, KENNETH JANSEN, ECAE, University of Colorado at Boulder — The interaction of a finite-span synthetic jet with the flow over a finite and swept back wing at a Reynolds number of \(10^6\) and at low angles of attack is studied by means of parallel adaptive flow simulations. The focus of the work is to explore the details of the flow structures in the vicinity of the synthetic jet, in coordination with experimental studies. Both instantaneous and phase-averaged flow fields are collected for that purpose. It is found that an array of counter-rotating vortical structures formed by the synthetic jet interacts with the cross flow, and develops three-dimensionalities as they are advected downstream. The effect of two blowing ratios (of 0.8 and 1.2) is also explored. In the case of low blowing ratio, coherent vortical structures are found to be dominant. At high blowing ratio, coherent vortical structures breakdown forming random ones. Finally, the predictions of the CFD simulations and the experimental measurements are compared.

\(^1\)The AFOSR is acknowledged for its support. The first author also acknowledges the BAEF.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GK Granular Flows 1 Long Beach Convention Center 201B

8:00AM GK.00001 Transient rheological behavior of suspensions near the jamming transition, ERIC BROWN, HEINRICH JAEGER, The University of Chicago — We performed transient rheological measurements on suspensions at several packing fractions near the jamming transition. A slow shear displacement was applied then, the shear stress was abruptly set to zero to observe the relaxation behavior. A harmonic oscillator model can be used to obtain the elastic part of the yield stress from oscillations and a transient viscosity from the relaxation time. For displacements smaller than a particle size elastic behavior is found if there is shear thinning at packing fractions below the jamming transition. For larger displacements, there is a relaxation but no oscillation. Remarkably, the transient viscosity differs from the steady-state viscosity at packing fractions near the jamming transition; the transient viscosity remains finite while the steady-state viscosity diverges at jamming.

8:13AM GK.00002 Granular collapse in a fluid: Role of the initial volume fraction, LOIC RONDON, OLIVIER POULQUIEN, PASCALE AUSSILLOUS, Laboratoire IUSTI, UMR 6595 CNRS, Aix Marseille Université (UI, UII), 5 rue Enrico Fermi, 13465 Marseille cedex 13, France, GROUPE ECOULEMENT DE PARTICULES TEAM — The collapse of a granular column suddenly released on a plane has been intensively studied the last ten years in the case of a granular medium with no interaction with the interstitial fluid. However, in many geophysical events like submarine avalanches or landslides, the interaction between the grains and the surrounding fluid plays a crucial role. In this work, we experimentally studied the collapse of a granular mass in a viscous fluid. We found that contrary to the dry case, the aspect ratio of the initial mass is no longer the only relevant parameter controlling the deposit morphology. In the viscous regime, the dynamics is controlled by the initial volume fraction of the mass. Two different regimes are identified. For initially loose packing the deposits are thin and long, the dynamics is fast and a positive liquid pressure is measured below the column. For dense packing, the run out distance is twice less, the flow is slow and a negative pore pressure is measured during the flow. These results suggest that the dynamics of the granular collapse in a fluid is strongly affected by the dilatancy or contractancy behaviour of the granular media.

8:26AM GK.00003 The effect of volume fraction on granular slope stability, NICK GRAVISH, School of Physics Georgia Tech, NICK WARD, Augsburg College, DANIEL I. GOLDMAN, Georgia Tech — We study the stability of granular slopes as a function of the prepared volume fraction 0.58 < \( \phi \) < 0.62. A bed of 250\( \mu \)m diameter glass beads with an initial slope angle \( \theta = 0^\circ \) and initial \( \phi \) is slowly rotated at constant angular velocity to a final angle of 30\(^\circ\). We monitor the motion of grains at the top surface and observe that the angle at which continuous surface flow occurs is sensitive to \( \phi \) and increases from \( \theta_0 \approx 26^\circ \) at low \( \phi \) (loosely packed) to \( \theta_0 \approx 32^\circ \) at high \( \phi \) (closed packed). Prior to the uniform failure at \( \theta_0(\phi) \) the grain motion during tilting differs between the loosely packed to the closely packed regimes. Tilting loosely packed beds results in rapid intermittent grain rearrangement at the surface; the angle at which these begin is \( \theta \approx 15^\circ \). In the closely packed beds grain motion at the surface is not observed until \( \theta \approx 31^\circ \), prior to continuous failure.

8:39AM GK.00004 Terminal velocity of a heavy object in a superlight granular medium, GABRIEL A. CABALLERO-ROBLEDO, Cimav, Unidad Monterrey, FELIPE PACHECO-VAZQUEZ, Cinvestav-IPN, Unidad Merida, J. CARLOS RUIZ-SUAREZ, Cinvestav-IPN, Unidad Monterrey — A granular material is a system composed of many solid particles interacting mainly through contact forces. Therefore, the dissipation of energy usually plays a dominant role in the dynamics of these materials. For this reason, in experiments done so far, when an object impacts on a granular bed it eventually dissipates all its energy and comes to rest. In contrast, when a dense enough object is placed inside a fluid it keeps falling, asymptotically approaching a terminal velocity. Here we present experiments of a heavy object falling into a silo full of expanded polystyrene spherical particles. The density of the granular medium is so low that it can not bear the weight of an intruder whose mass is beyond a threshold value, even if the object is very deep in the silo. We systematically vary the mass of an object impacting in such a granular bed and we find a transition between the commonly observed behavior where the object stops at a given depth, and a situation where the object keeps falling and reaches a terminal velocity, just as in a fluid.

8:52AM GK.00005 ABSTRACT WITHDRAWN —

9:05AM GK.00006 Stokes’ Cradle: Oblique Collisions between Wetted Particles, CARLY DONAHUE, CHRISTINE HRENYA, ROBERT DAVIS, WILLIAM BREWER, University of Colorado — Granular particles can be made more cohesive by applying a viscous liquid to the surface of the particle. Such wetted particles are naturally involved in pollen capture and avalanches and can be found in industrial processes such as granulation and filtration. The focus here is on collisions between wetted particles in which lubrication forces dominate over capillary forces (i.e., high capillary number). Previous experiments with such systems have been limited to normal (head-on) collisions of spheres and collisions between a sphere and an immobile wall. In these cases, rebound (de-agglomeration) was found to depend upon the surface roughness of the solids, the elastohydrodynamic interaction, or the pressure-dependent viscosity. In this effort, we experimentally investigate collisions between two wetted particles impacting at an oblique angle. Now, in addition to the above interactions, the presence of a centrifugal force also contributes to the mechanism for rebound-agglomeration. A theoretical analysis of the associated regime maps provides useful insight to unravel the relevant physical processes that occur in oblique collisions.

9:18AM GK.00007 Macroscopic Characteristics of Unsteady Granular Flows in Rotating Tumblers, DANIEL PAPROCKI, Northern Illinois University, NICHOLAS POHLMAN, Northern Illinois University — Flow of silicate beads in rotating tumblers of triangular cross-sections are explored with respect to transient response of macroscopic properties. High-speed digital images are synchronized to tumbler orientation through an in-line rotary encoder. Image processing toolboxes are utilized to generate quantitative data for analysis. Time-dependent properties of free surface length, flowing layer curvature, and dynamic angle of repose are reported. The correlation of these properties with the orientation exhibits a phase difference that is a function of tumbler dimensions and fill fraction. Concurrent measurements of input energy to the system may lead to control algorithms to generate steady flow in inherently unsteady systems that would improve efficiency of granular transport methods.
9:31AM GK.00008 A comparison of the granular flow of glass spheres and sands, STEVEN W. MEIER, KATHERINE P. BARTEAU, DENIZ ERTAS, HUBERT E. KING, ExxonMobil Research and Engineering — We investigate the effect of particle shape irregularity on granular flow on an erodible bed 2 cm wide confined by sidewalls 50 cm long and 20 cm tall. Three types of particles were studied: 0.65 mm spherical glass particles, round sand particles with a mean size of 0.87 mm, and angular sand particles with a mean size of 0.83 mm. The dynamics of the flows were measured with a high speed camera. For all particle types, the angle between the flowing free surface and horizontal increases linearly with mass flow rate. While the surface angle is greater for flows of sand particles than spherical glass particles at all mass flow rates, the change in surface angle with respect to mass flow rate is smaller for flows of sand than spherical glass particles. For all particle types, the velocity profile increases exponentially into the flow layer from the erodible bed. For fast flows, the velocity profile is linear in the upper portion of the flow layer. However, spherical glass particles flow in thicker flowing layers with lower surface velocities than sand particles for comparable mass flow rates.

9:44AM GK.00009 Cutting and shuffling—Mixing in Spherical Tumblers, RICHARD M. LUEPTOW, IVAN C. CHRISTOV, Northwestern University, GABRIEL JUAREZ, University of Pennsylvania, JULIO M. OTTINO, Northwestern University, ROB STURMAN, University of Leeds, STEPHEN WIGGINS, University of Bristol — Cutting and shuffling and the underlying mathematical formalism of piecewise isometries (PWIs) offer means to predict mixing of granular material in 3D tumblers. We have studied various mixing protocols for two-axis rotation of a spherical tumbler using a continuum model with a vanishingly-thin flowing layer (PWIs) and a realistic-thickness flowing layer. The PWIs describe the skeleton of the mixing emerging from the container shape, fill fraction, and rotation protocol. Mixing measures based on the center of mass of an ensemble of seed tracer particles and based on concentration variance provide similar results. Poor mixing occurs when the center of mass of the seed particles does not converge to the center of mass of the domain, but instead evolves toward a “limit cycle” due to symmetries in the rotation protocol. Good mixing occurs when the “limit cycle” is avoided. Comparison with simulations having a realistic flowing layer discerns the role of the flowing layer’s thickness on mixing. The quality of mixing predicted by the center of mass measure based on the model with a vanishingly-thin flowing layer (PWIs) correlates with the quality of mixing predicted by the decay of concentration variance, allowing for fast optimization of mixing protocols using PWIs.

9:57AM GK.00010 Particle Tracking Velocimetry and Granular Flow Correlations in Triangular Tumblers, JOSEPH SZALIKO, NICHOLAS Pohlman, Northern Illinois University — Granular flow has traditionally been examined at steady state with time averaged results. Circular shaped tumblers with constant rotation rates eliminate most transient effects in dynamic flow. This research examines the transient flow induced by triangular shaped tumblers. High speed imaging and custom particle tracking velocimetry (PTV) are used to analyze several aspects of the flow: shear layer thickness along the variable angle of repose, transverse flow within the shear layer, and velocity profiles at different tumblers orientations and dimensions. Correlations of these properties with one another and the time/orientation dependence of the non-uniform tumbler are reported. Results indicate transient flow may not be equivalent to instantaneous conditions of steady flow. For example, highest velocities exist where the shear layer is thinnest.

Monday, November 22, 2010 8:00AM - 10:10AM —
Session GL Biofluids: Physiological Circulatory I Long Beach Convention Center 202A

8:00AM GL.00001 Stability of two-dimensional collapsible-channel flow to inviscid perturbations, TIMOTHY PEDLEY, University of Cambridge, REMESH KUDENATTI, Bangalore University — We consider the linear stability of two-dimensional inviscid but vortical flow in a rigid, parallel-sided channel, of which part of one wall has been replaced by a flexible membrane under longitudinal tension $T$. Far upstream the flow is parallel Poiseuille flow; the width of the channel is $a$ and the length of the membrane is $\lambda a$, where $\lambda \gg 1$. Steady flow at high Reynolds number $Re$ was studied using interactive boundary-layer theory by Guneratne & Pedley (J. Fluid Mech. 569, 151-184, 2006) for various values of the pressure difference $P_2$ across the membrane at its upstream end. Here we study small-amplitude, unsteady perturbations to the trivial steady solution for $P_2 = 0$. An unexpected finding is that the flow is always unstable, with a growth rate that increases with $T$. In other words, the stability problem is ill-posed. However, when the pressure difference is held fixed (= 0) at the downstream end of the membrane, the problem is well-posed and all solutions are stable. The physical mechanisms underlying these findings are explored; the crucial factor in the fluid dynamics is the vorticity gradient across the incoming Poiseuille flow. Similar results are found for the viscous problem at high $Re$.

8:13AM GL.00002 Non-homogeneous concentration of suspensions in micro-capillary networks: particles in a bifurcation, THOMAS PODGORSKI, CNRS-UJF Grenoble, VINCENT DOYEUX, UJF Grenoble, SARAH PEPONAS, Université de Picardie, MOURAD ISMAIL, UJF Grenoble, GWENNOUN COUPIER, CNRS-UJF Grenoble — We investigate the distribution of particles in flows of dilute suspensions in bifurcating channels. In studies relevant to blood flow in the microcirculation, an increase of the volume fraction of particles (hematocrit) in the high flow rate branch is usually observed, leading to non-uniform concentrations in a network of channels, with possible consequences on oxygen transport and pressure distribution. In the literature, this phenomenon is often wrongly interpreted as the result of some attraction of the particles towards this high flow rate branch. We show thanks to experiments and numerical simulations that the concentration phenomenon, often referred to as Zweifach-Fung effect, is mainly due to the non-homogeneous spatial distribution of particles in the mother branch, while a weak attraction towards the low flow rate branch occurs in the bifurcation.

8:26AM GL.00003 A dynamic model of human physiology1, MELISSA GREEN, CAROLYN KAPLAN, ELAINE ORAN, JAY BORIS, Naval Research Laboratory — To study the systems-level transport in the human body, we develop the Computational Man (CMAN): a set of one-dimensional unsteady elastic flow simulations created to model a variety of coupled physiological systems including the circulatory, respiratory, excretory, and lymphatic systems. The model systems are collapsed from three spatial dimensions and time to one spatial dimension and time by assuming axisymmetric vessel geometry and a parabolic velocity profile across the cylindrical vessels. To model the actions of a beating heart or expanding lungs, the flow is driven by user-defined changes to the equilibrium areas of the elastic vessels. The equations are then iteratively solved for pressure, area, and average velocity. The model is augmented with valves and contractions to resemble the biological structure of the different systems. CMAN will be used to track material transport throughout the human body for diagnostic and predictive purposes. Parameters will be adjustable to match those of individual patients. Validation of CMAN has used both higher-dimensional simulations of similar geometries and benchmark measurement from medical literature.

1This work was supported, in part, by the NRC Research Associate Program, ONR through NRL, and DTRA-JSTO for Chemical and Biological Defense.
the influence of the background flow, cluster size, magnetic field strength and Brownian motion. We derive simplified estimates for the achievement of optimal vessels. An external magnetic field is applied in order to attract the cluster toward a prescribed target. The transit time is obtained numerically and assessed for

Tech, SHAHRIAR AFKHAMI, New Jersy Institute of Technology, YURIKO RENARDY, Virginia Tech — The motion of a superparamagnetic hydrophobic clusters in a pressure-driven channel flow with an external magnet

Vessels for Focused Therapy

collectors within the blood vessel by an externally applied magnet. The capture of maghemite nanoparticles was studied in blood vessels as a function of fluid velocity, vessel diameter, magnetic field strength and fluid viscosity. Nanoparticles were captured most easily in small blood vessels with applications of higher magnetic fields. Higher viscosity fluids cause a reduction in the effective capture of nanoparticles. By studying localization in water and simulated blood plasma, the importance of studying flow behavior in complex fluids for further development of medical therapies is evident.

9:05AM GL.00006 Magnetic Localization of Maghemite Nanoparticles in Simulated Blood Vessels for Focused Therapy3, NATALIE LAP, Manhattan College, CHRISTOPHER BRAZEL, The University of Alabama — Magnetic nanoparticles (MNPs) can easily be administered to patients intravenously for use in therapies such as hyperthermia or localized drug delivery. The MNPs are collected within the blood vessel by an externally applied magnet. The capture of maghemite nanoparticles was studied in blood vessels as a function of fluid velocity, vessel diameter, magnetic field strength and fluid viscosity. Nanoparticles were captured most easily in small blood vessels with applications of higher magnetic fields. Higher viscosity fluids cause a reduction in the effective capture of nanoparticles. By studying localization in water and simulated blood plasma, the importance of studying flow behavior in complex fluids for further development of medical therapies is evident.

9:31AM GL.00008 Blood flow and blood cell interactions and migration in microvessels. DMITRY FEDOSOV, JULIA FORNLEITNER, Postdoc, GERHARD GOMPPER, Professor — Blood flow in microcirculation plays a fundamental role in a wide range of physiological processes and pathologies in the organism. To understand and, if necessary, manipulate the course of these processes it is essential to investigate blood flow under realistic conditions including deformability of blood cells, their interactions, and behavior in the complex microvascular network which is characteristic for the microcirculation. We employ the Dissipative Particle Dynamics method to model blood as a suspension of deformable cells represented by a viscoelastic spring-network which incorporates appropriate mechanical and rheological cell-membrane properties. Blood flow is investigated in idealized (e.g., channels, tubes) and complex (e.g., expansions, bifurcations) geometries. In particular, migration of blood cells and their distribution in blood flow are studied with respect to various conditions such as hematocrit, flow rate, red blood cell aggregation, and vessel geometry. Physical mechanisms which govern cell migration in microcirculation and, in particular, marginalization of white blood cells towards the vessel wall, will be discussed. In addition, we characterize blood flow dynamics and quantify hemodynamic resistance in the microvascular network.

9:44AM GL.00009 Hydrodynamic forces on a wall-bound leucocyte in small vessels due to red cells, AMIR H. G. ISFAHANI, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — As part of the inflammation response, white blood cells (leukocytes) bind to the vessel wall before they transmigrate across the endothelium. The interactions between the wall-adhered leukocyte and flowing red blood cells (erythrocytes) play a critical role in this process. We provide a quantitative investigation of the forces exerted on a wall-bound leucocyte using a simulation tool that is based on a fast $O(N \log N)$ boundary integral formulation. This permits simulations of red cells that are both realistically flexible and can approach to very close separation distances. The elastic membranes deform substantially but strongly resist surface dilatation. The no-slip condition is enforced both on the leucocyte and the round vessel walls. Vessel diameters from 10 to 20 microns are studied. At these scales the cellular-particulate nature of blood significantly affects the magnitude of the forces that the leucocyte experiences. For a tube hematocrit (cell volume fraction) of 25% and a spherical protrusion with a diameter 0.75 times that of the tube, the average forces are increased by about 40% and the local forces by more than 100% relative to those expected for a blood model homogenized by its effective viscosity. For a constant pressure gradient, the wall-bound leucocyte causes a blockage in the vessel. Different contact angles for the leucocyte as well as different mechanical properties for the erythrocytes are examined.

9:57AM GL.00010 Evaluation of RBC aggregation using synchrotron X-ray speckles1, HOJIN HA, KWON-HO NAM, SANG JOON LEE, POSTECH, BBRC TEAM — When a coherent beam illuminates spatially-disordered particles, speckles are usually generated by the interference of the scattered light waves. The speckle has been known to contain the information of the objects under near-field condition. In this study, we hypothesized that the speckle patterns of the red blood cells are related to the aggregation shape and the size of RBCs in the medium. The speckle patterns of RBCs in static condition were investigated by transmitting the monochromatic synchrotron X-ray beam to the sample with varying hematocrit(10-80%) and medium type(phosphate buffered saline, autologous plasma and 0.75 % polyvinylpyrrolidone 360 in phosphate buffered saline). The temporal variation of speckle patterns after sudden removal of shear rate was observed by stopping the blood flow in a tube. The size of aggregated RBCs is closely correlated with the characteristic features of the speckle patterns.

1This work was supported by the Creative Research Initiatives (Diagnosis of Biofluid Flow Phenomena and Biomimic Research) of MEST/NRF of Korea.

X-ray imaging experiments were performed at 1B2 and 7B2 beamlines of Pohang Accelerator Laboratory (PAL), KOREA

Monday, November 22, 2010 8:00AM - 10:10AM – Session GM Microfluids: General IV: Bubbles, Drops and Particles Long Beach Convention Center 202B
8:00AM GM.00001 Wettability dependent bubble dynamics in microfluidic networks, PRAVIEN PARTHIBAN, SAIF A. KHAN, National University of Singapore, MICHAEL T. KREUTZER, Delft University of Technology — The routing of bubble or droplet traffic through microfluidic networks depends greatly on the hydrodynamic resistance in the individual branches of that network. We find that a confined bubble translating through a partially wetting liquid experiences significantly more friction than a bubble lubricated by a completely wetting liquid, with important consequences for the dynamic behavior. For our system, we observe symmetric bubble break up and alternating left-right routing at a microfluidic junction, as described previously by Link et al. For partially wetting liquids, we observe a much richer dynamic behavior, with asymmetric splitting and left-right routing with chaotic periodicity. We identify the contact angle as a key control parameter that determines the different regimes and we explore how the transitions between these regimes can be effected by tuning this parameter. The results of this work aid the prediction and control of bubble traffic through complex microfluidic networks. Link et al., Phys. Rev. Lett. 92 (2005) 054503

8:13AM GM.00002 Flow field inside a stationary microdroplet in a Hele-Shaw cell, SUNGYONG LEE, CHARLES BAROUD, LadHyX and department of Mechanics, Ecole Polytechnique — We consider the flow field inside a water drop held stationary in a flowing external oil, experimentally and theoretically. The droplet is strongly confined in the vertical direction, making it take a “pancake”-like shape. It is anchored in place by inducing a local variation in the channel height which reduces the free surface energy with minimal modifications to its general shape. Two contrasting flow regions are visible inside the drop: a fast recirculation flow is observed near the droplet boundary, while a slower flow takes place in the central region. While the central flow is well described by the standard Hele-Shaw model, the flow near the droplet edge displays strong three-dimensional recirculation, pointing to a complex hydrodynamic coupling between the droplet and outer flows. These two regimes are characterized for different droplet geometries and external flow rates, and a theoretical justification for their existence is provided.

8:26AM GM.00003 The Microfluidic Thunderstorm, ALVARO G. MARIN, WIM VAN HOEVE, Physics of Fluids, Univ. Twente, LINGLING SHUI, JAN C.T. EIJKEL, ALBERT VAN DEN BERG, Bios/lab-on-chip group, Univ. Twente, DETLEF LOHSE, Physics of Fluids, Univ. Twente — The so-called “Kelvin’s thunderstorm” is a simple experiment demonstrating the spontaneous appearance of induced free charge in droplets emitted through a tube. As Lord Kelvin explained, the droplets acquire a net charge spontaneously during pinch-off due to the presence of electrical fields in their surrounding created by any metallic object. In his experiment, two streams of droplets are allowed to drip from separated nozzles into separated buckets, which are at the same time interconnected through the dripping needles. The implementation of such an effect in a microfluidic device could enhance the control of droplets and prevent undesired effects as coalescence. The phenomenon has been successfully reproduced in a simple microfluidic device, where the droplets could get charged to charge-to-mass ratios above the Rayleigh limit. Experimental measurements will be presented showing the dependence of the acquired charge on the droplets on different parameters as the flow rate or the liquid electrical conductivity. The concept certainly opens a door to a costless and accessible transformation of pneumatic pressure into electrical energy and to an enhanced control in microfluidic technologies.

8:39AM GM.00004 Droplet flows through periodic loop networks, RAPHAEL JEANNERET, MICHAEL SCHINDLER, DENIS BARTOLO, ESPCI — Numerous microfluidic experiments have revealed non-trivial traffic dynamics when droplets flow through a channel including a single loop. A complex encoding of the time intervals between the droplets is achieved by the binary choices they make as they enter the loop. Very surprisingly, another set of experiments has demonstrated that the addition of a second loop does not increase the complexity of the droplet pattern. Conversely, the second loop decodes the temporal signal encrypted by the first loop [1]. In this talk we show that no first principle argument based on symmetry or conservation laws can account for this unexpected decoding process. Then, to better understand how a loop maps time intervals between droplets, we consider a simplified model which has proven to describe accurately microfluidic droplet flows. Combining numerical simulations and analytical calculations for the dynamical behavior of droplets circulating through the two loops, we will argue that none of them yields exact decoding. (i) We uncover that for a wide class of loop geometry, the coding process is analogous to a Hamiltonian mapping: regular orbits are destabilized in island chains and separatrix. (ii) Eventually, we propose a simple explanation to solve the apparent paradox with the coding/decoding dynamics observed in experiments. [1] M.J. Fuerstman, P. Garstecki, and G.M. Whitesides, Science, 315:828, 2007.

8:52AM GM.00005 Patterning of non-spherical particles onto electrode surface: Study of orientation behavior under viscous fluid and AC electrokinetic forces, RAVIRAJ THAKUR, Purdue University, STUART WILLIAMS, ROBERT COHN, JEREMY RATHFON, University of Louisville, JEAN-FRANÇOIS BERRET, MINHAO YAN, Université Denis Diderot-Paris, STEVEN WERELEY, Purdue — Recently we had proposed a technique called rapid electrokinetic patterning(REP), a tool that can manipulate colloidal particles near illuminated spot on an electrode surface. REP utilizes optical landscapes to create gradients in temperature allowing local changes in permittivity and conductivity of the fluid creating a microvortex. However, REP has been demonstrated till now only with spherical particles. We expand upon the initial disclosure of REP and conduct experiments with non-spherical beads. In the presence of linearly polarized field a non-spherical particle experiences frequency dependent alignment torques along three principle axis. This is mainly because of the different polarizability along each direction. In a fluid flow, a non-spherical particle would align itself in order to minimize the viscous drag. But characterizing the orientation behavior of non-spherical particles under the influence of both electric field and viscous drag presents a unique physics problem. We observe the vertical orientation of the cylinders in the REP aggregation. We explore the mobility of the captured particles on the surface with respect to various physical parameters.

9:05AM GM.00006 Particles Dispersion on Fluid-Liquid Interfaces, PUSHPENDRA SINGH, SATHISH GURU-PATHAM, BHAVIN DALAL, M. HOSSAIN, IAN FISCHER, NJIT, D.D. JOSEPH, University of Minnesota — In a previous study we have shown that when small particles, e.g., flour, pollen, etc., come in contact with an air-liquid interface, they disperse in a manner that appears explosive. This is due to the fact that the capillary force pulls particles into the interface causing them to accelerate to a relatively large velocity. The motion of particles in the direction normal to the interface is inertia dominated, and so they oscillate vertically about the equilibrium position before coming to a stop under viscous drag. This vertical motion of the particles causes a local variation in the channel height which reduces the free surface energy with minimal modifications to its general shape.

9:18AM GM.00007 Microfluidic enhanced conductive polymer microspheres for sensor applications, JESSICA SNYDER, University of Washington, DONGLAI LU, Pacific Northwest Laboratory, AMY SHEN, University of Washington — Methods and devices were developed to produce monodisperse, conducting, responsive polyaniline (PANI) particles for drug delivery and sensor applications. Liquid droplets are produced containing a dispersed phase carried through the device by the continuous phase. The two phases are immiscible. Each phase can be either oil or water based. The aniline monomer is contained within the dispersed phase while the oxidizing agent, ammonium persulfate (APS) is contained within the aqueous phase. The production of either solid (aniline, APS in dispersed phase) or shell particles (aniline in dispersed phase, APS in continuous phase) is possible. Droplets are formed by controlling the viscous and capillary forces at the interface. Droplet size is controlled by phase flow rates, the interfacial tension and viscosity ratio between the fluid phases and the inlet geometry. PANI particles are produce via oxidative polymerization. The polymerization is pH dependent and the time of polymerization is monitored by the distance the droplets travel in the channel. The morphology and electrochemical characteristics of the particles resulting from these methods are studied.
9:31AM GM.00008 Transport and separation of micron sized particles at isotachophoresis zone boundaries , STEFFEN HARDT, GABRIELE GOET, TOBIAS BAIER, Institute for Nano- and Microfluidics, Center of Smart Interfaces, TU Darmstadt — Conventionally, isotachophoresis (ITP) is used for the separation of ionic samples according to their electrophoretic mobilities. At the zone boundaries large gradients in concentration and electric field occur. These gradients may be utilized to transport and separate small particles, as we demonstrate experimentally. We show that polymer beads of 5 micron diameter dispersed in a high mobility leading electrolyte are picked up and carried along by an ITP zone boundary that is formed between a low mobility trailing electrolyte and the leading electrolyte. Additionally, it is shown that different types of beads can be separated in that way. In particular, beads of 1 micron diameter are not carried along by the transition zone, so that a separation from 5 micron sized beads is feasible. We have identified two different effects that contribute to the force acting on the particles. Firstly, there is an electric dipole force due to the electric field gradient, secondly, a hydro electrostatic force is generated that induces a pressure gradient. Therefore, the resulting protocol for particle separation bears some resemblance with dielectrophoresis that also utilizes electric dipole forces. An apparent advantage of our technique over dielectrophoresis lies in the fact that no microstructured electrodes or other types of microstructures are needed to create the electric field gradient.

9:44AM GM.00009 3D inertial migration of microspheres in a microchannel flow: transition from radial to angular focusing1, YONG-SEOK CHOI, Biofluid and Biomimetic Research Center, Department of Mechanical Engineering, POSTECH, Korea, SANG-JOON LEE, Biofluid and Biomimetic Research Center, Department of Mechanical Engineering, School of Interdisciplinary Bioscience and Bioengineering, POSTECH, Korea — The “2P” mode vortex wake, in which two vortex pairs are generated per shedding cycle, is a commonly occurring wake structure behind oscillating bluff bodies. We will present an idealized model of these wakes that consists of a singly-periodic Hamiltonian system of four point vortices. The system is made integrable with an imposed spatial symmetry that is motivated by constraints. The method is applied to two-dimensional vortex dynamics and a variety of numerical examples are given, including the calculation of monopole, planar, and icosahedral relative equilibria and lead to a problem of finding solutions to equations for interparticle distances. Obtained equations give sufficient conditions for the relative equilibria and lead to a problem of finding solutions to $A\Gamma = 0$, where $\Gamma \in R^M$ is the vector of vortex strengths, and $A \in R^{M \times N}$ is a rectangular, non-normal ($A^T A \neq A A^T$) “configuration” matrix determined by the particle positions. Using singular value decomposition of $A$ we prove that for icoshedron the $Nullspace(A)$ is 7 dimensional. Vertex and edge stabilizers, as subgroups of icoshedral symmetry group, are used to build the set of symmetric icoshedral configurations with non-negative strengths. Using exact solution of equations of motion we prove stability of vortex pair configurations. Energy-momentum method is used to study stability of symmetric icoshedral relative equilibria. To prove instability of some of the configurations we show that the matrix of linearized system has eigenvalues with positive real parts. Using the stability results we build an example of linear superposition of stable configurations which gives unstable configuration.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GN Vortex Flows: Inviscid Long Beach Convention Center 202C

8:00AM GN.00001 Relative equilibria of point vortices and the fundamental theorem of algebra1, HASSAN AREF, DTU and Virginia Tech — The fundamental theorem of algebra implies that every non-zero single-variable polynomial with complex coefficients has exactly as many complex roots as its degree, if each root is counted with its multiplicity. This result may be applied to the generating polynomial for a relative equilibrium of point vortices and used to derive differential equations for this polynomial in various situations, e.g., when the vortices are on a line or all on a circle. The derivations thus obtained are quite elegant and compact compared to the corresponding derivations found in the literature. A new formula that provides the basis for application of the fundamental theorem to vortex equilibria is outlined and a number of the further derivations demonstrated.

1Supported by the Danish National Research Foundation through a Niels Bohr Visiting Professorship.

8:13AM GN.00002 Hamiltonian-Dirac Simulated Annealing: Application to the Calculation of Vortex States , P.J. MORRISON, The University of Texas at Austin, G.R. FLIERL, Massachusetts Institute of Technology — A simulated annealing method for calculating stationary states for models that describe continuous media is proposed. The method is based on the noncanonical Poisson bracket formulation of media, which is used to construct Dirac brackets with desired constraints, and symmetric brackets that cause relaxation with the desired rate. The method is applied to two-dimensional vortex dynamics and a variety of numerical examples are given, including the calculation of monopole and dipole vortex states.

8:26AM GN.00003 Mathematical modeling of “2P” mode vortex wakes , SAIKAT BASU, MARK STREMLER, Virginia Tech, TEIS SCHNIPPER, ANDERS ANDERSEN, Technical University of Denmark — The “2P” mode vortex wake, in which two vortex pairs are generated per shedding cycle, is a commonly occurring wake structure behind oscillating bluff bodies. We will present an idealized model of these wakes that consists of a singly-periodic Hamiltonian system of four point vortices. The system is made integrable with an imposed spatial symmetry that is motivated by constraints. The method is applied to two-dimensional vortex dynamics and a variety of numerical examples are given, including the calculation of monopole and dipole vortex states.

8:39AM GN.00004 Stability of icoshedral configurations of point vortices on a sphere , VITALII OSTROVSKYI, PAUL NEWTON, University of Southern California — Using icoshedron as the initial geometric distribution of point vortices on a sphere we show existence of icoshedral relative equilibrium configurations. To characterize these configurations we apply method based on finding the fixed points of the nonlinear dynamical system governing the $N(N - 1)/2$ equations for interparticle distances. Obtained equations give sufficient conditions for the relative equilibria and lead to a problem of finding solutions to $A\Gamma = 0$, where $\Gamma \in R^M$ is the vector of vortex strengths, and $A \in R^{M \times N}$ is a rectangular, non-normal ($A^T A \neq A A^T$) “configuration” matrix determined by the particle positions. Using singular value decomposition of $A$ we prove that for icoshedron the $Nullspace(A)$ is 7 dimensional. Vertex and edge stabilizers, as subgroups of icoshedral symmetry group, are used to build the set of symmetric icoshedral configurations with non-negative strengths. Using exact solution of equations of motion we prove stability of vortex pair configurations. Energy-momentum method is used to study stability of symmetric icoshedral relative equilibria. To prove instability of some of the configurations we show that the matrix of linearized system has eigenvalues with positive real parts. Using the stability results we build an example of linear superposition of stable configurations which gives unstable configuration.
8:52AM GN.00005 The motion of singularities in potential flow, STEFAN LLEWELLYN SMITH, MAE, UCSD — In the first paper on vorticity, Helmholtz discussed infinitesimal rectilinear filaments, and Kirchhoff subsequently derived the equation of motion of point vortices. This equation can be viewed as the statement that the translational velocity of the point vortex is obtained by removing the leading-order singularity due to the point vortex when computing its velocity. I review the arguments used to obtain this result and discuss their history and limitations. I then examine the extension of these ideas to other kinds of singularities and give some examples.

9:05AM GN.00006 An accurate and efficient method to compute steady uniform vortices, P. LUZZATTO-FEGIZ, C.H.K. WILLIAMSON, Cornell University — Steady uniform vortices represent a widely used approximation in a broad range of contexts, ranging from dynamics of plasmas to geophysical flows. Surprisingly, computing steady uniform vortices still presents several challenges, since vortex boundaries may develop high-curvature regions, which can be prohibitively expensive to resolve. Further to this, flows can bifurcate to lower-symmetry states, which may be difficult to compute reliably. Currently, one must choose between affordable relaxation methods and more reliable approaches based on Newton iteration. However, while the first cannot resolve flows without symmetry (Dritschel 1985), the second are unaffordable for vortices with high-curvature regions (Saffman & Szeto 1980; Elcrat et al. 2005). Hence it is typically impossible to compute a family of steady vortices in its entirety. We overcome these limitations by introducing a new discretization, based on an “inverse-velocity map”, which makes Newton iteration affordable for vortices with high-curvature boundaries. By employing our numerical method in conjunction with the IV-diagram stability approach (L&W PRL 2010), we explore the full bifurcation structure of several classical flows, including elliptical vortices, co-rotating and countercorotating vortex pairs, and vortex streets. We have also successfully employed our discretization for other fluid problems, such as steep gravity waves.

9:18AM GN.00007 A Framework for Linear Stability Analysis of Finite–Area Vortices, BARTOSZ PROTAS, McMaster University, ALAN ELCRAT, Wichita State University — In this work we are interested in the linear stability of 2D solutions of the Euler equations which are steady in the appropriate frame of reference and feature compact regions with constant vorticity embedded in an otherwise potential flow. We argue that, since the evolution of such systems is governed by equations of the free–boundary type, the shape calculus is a natural framework for differentiating the solutions. As a result, a general equation characterizing the evolution of area–preserving perturbations of the boundary. While for vortex regions with arbitrary shapes the perturbation equation needs to solved numerically (e.g., using spectral Fourier–Galerkin method), we show that for a circular boundary (i.e., the Rankine vortex) the problem can be solved analytically yielding the classical stability results due to Kelvin. We will also present stability calculation obtained numerically for more general vortex shapes and will discuss generalizations of this approach.

9:31AM GN.00008 Instability of Point Vortex Leapfrogging, LAUST TOPHØJ, Department of Physics, Technical University of Denmark, HASSAN AREF, ESM Dept., Virginia Tech, and Center for Fluid Dynamics, Technical University of Denmark — The dynamics of interacting point vortices on the unbounded plane can be chaotic if the number of vortices is at least four. The chaotic dynamics is governed by the existence of unstable structures in the phase space, [Tophøj & Aref, Phys. Fluids, 20, 093605 (2008)]. Such structures may be hyperbolic fixed points of the dynamical system, or unstable periodic orbits. Chaos arises as the system is repeatedly repelled by these structures, bouncing back and forth between them. The leapfrogging motion of two vortex pairs possessing a common axis of symmetry is an example of an integrable periodic motion of a four-vortex system. The stability of this periodic motion has been studied numerically by Acheson [Eur. J. Phys. 21, 269 (2000)] whose results indicate instability for some but not all parameters. We discuss the stability of leapfrogging, using methods from Floquet theory. Analogies will be drawn to instabilities of the von Kármán vortex street that can cause the vortex street to break up into vortex pairs moving away from the central axis.

Supported in part by Danish National Research Foundation.

9:44AM GN.00009 Lagrangian trajectories in Lissajous vortices, SERGIO CUEVAS, ALDO FIGUEROA, EDUARDO RAMOS, Centro de Investigacion en Energia, Universidad Nacional Autonoma de Mexico — We report Particle Image Velocimetry experiments in a rectangular container with a shallow layer of an electrolyte in which a vortex flow is driven by Lorentz forces produced by the field of a permanent cylindrical magnet and two alternate electric currents perpendicular to each other. Currents are injected through two pairs of parallel electrodes located at the container walls but avoiding short circuit. Due to the harmonic forcing in perpendicular directions, the system is excited analogously to the kinematic Lissajous figures although in the fluid case convective and viscous effects are present. In the creeping flow limit, an analytical solution is obtained so that the Lagrangian trajectories can be integrated. A full numerical solution that accounts for cases where non-linear effects are important is also used in the analysis. Lagrangian trajectories based on analytical, numerical and experimental results are compared for different values of amplitudes, frequencies and relative phases of the electromagnetic forcing.

Supported by CONACYT under project 59977.

9:57AM GN.00010 A Robust Numerical Method for Integration of Point-Vortex Trajectories in Two Dimensions, SPENCER SMITH, Tufts University Physics Department, BRUCE BOGHSOSIAN, Tufts University Department of Mathematics — The venerable 2D point-vortex model plays an important role as a simplified version of many disparate physical systems, including superfluids, Bose-Einstein condensates, certain plasma configurations, and inviscid turbulence. Point-vortex dynamics are described by a relatively simple system of nonlinear ODEs which can easily be integrated numerically using an appropriate adaptive time stepping method. As the separation between two vortices relative to all other inter-vortex length scales decreases, however, the computational time required diverges. Accuracy is usually the most discouraging casualty when trying to account for such vortex motion, though the varying energy of this ostensibly Hamiltonian system is a potentially more serious problem. We solve these problems by a series of coordinate transformations: We first transform to action-angle coordinates, which, to lowest order, treat the close pair as a single vortex amongst all others with an internal degree of freedom. We next, and most importantly, apply Lie transform perturbation theory to remove the higher-order correction terms in succession. The overall transformation drastically increases the numerical efficiency and ensures that the total energy remain constant to high accuracy.

Monday, November 22, 2010 8:00AM - 10:10AM –
Session GP Microfluids: Fluidic Devices — Long Beach Convention Center 203A

8:00AM GP.00001 Continuous Size-Based Particle Separation in a Microfluidic Device, BARUKYAH SHAPARENKO, HAN-SHENG CHUANG, HOWARD HU, HAIM BAU, University of Pennsylvania, GEORGE WORTHEN, Children’s Hospital of Philadelphia — Pinched flow fractionation is a continuous particle sorting technique in which two streams (one with particles, the other without particles) are manipulated to meet and then flow collinearly through a pinched microchannel. Due to geometric constraints, the particles align at different positions relative to the channel wall, with smaller particles closer to the wall than larger particles. Following the pinched segment, the channel broadens significantly, and the differences in particle positions are amplified as the particles follow the diverging fluid streamlines and are separated into different outlet channels based on their sizes. We analyze the separation of 2 and 10 µm rigid spherical particles with a pinched segment of 40 µm width, comparing 2D computational results and experimental results. We control the separation by specifying an inlet flow rate ratio and one outlet flow rate. We optimize the channel geometry and determine the operating parameters necessary to achieve effective particle separation. Multiple stages of such separation components can be integrated for finer separations. Other separation mechanisms, like dielectrophoresis, can also be integrated into the device using field flow fractionation, in which an external field is applied perpendicular to the direction of flow, causing the particles to cross fluid streamlines.
We also demonstrate that polystyrene beads can be continuously focused, trapped, concentrated, and separated in microfluidic reservoirs. This diverse electrical and simplicity. The success of these devices depends on a comprehensive understanding of electrokinetic particle transport in every part of their microchannels. We demonstrate a size-based separation of polystyrene beads (1 \( \mu \)m and 5 \( \mu \)m in diameter) and microbial cells (E. coli and yeast) in the serpentine microchannel with the application of a small DC electric field. We also develop a numerical model to simulate the particle and cell separation process.

Microchannel ridge arrays are widely used for biomedical research and clinical application. We design a microfluidic channel with periodically arranged diagonal ridges that separate and fractionate. Finally, we discuss how our numerical simulations may be of use in device prototyping and optimization.

8:26AM GP.00003 Long Chain DNA Separation in a Sparse Nanopost Array, JIA OU, MARK JOSWIAK, KEVIN DORFMAN, Chemical Engineering and Materials Science, University of Minnesota — Long chain DNA separation is a challenge for gel lectrophoresis. Our previous DNA separation experiments and simulations demonstrated that a sparse micro post array can separate large DNA. However, the smaller DNA are not well resolved. We hypothesized that smaller posts will increase the collision frequency of the smaller DNA and thus the resolution. We successfully fabricated a hexagonal array of 350 nm diameter posts with a 3 \( \mu \)m spacing using an oxygen plasma etching method. Under an electric field of 10 V/cm, the mobilities of different species ranging from 10-48.5 kilobasepair (kbp) were normalized by the mobility of \( \lambda \) DNA (48.5 kbp), which was included in all experiments as a standard to correct for day-to-day variations in electroosmotic flow. The resolution of these DNA is markedly improved when compared with a 1 \( \mu \)m diameter micropost array. We demonstrate the robustness of the device by using the calibration curve to identify the peaks in a separation of the \( \lambda \) DNA-Mono Cut mix.

This work was supported by the Singapore-MIT Alliance.

8:52AM GP.00005 Continuous separation of microparticles by size in ridged microchannels, WENBIN MAO, GONGHAO WANG, TODD A. SULCHEK, ALEXANDER ALEXEEV, Georgia Institute of Technology — Size-based separation and sorting are widely used for biomedical research and clinical application. We design a microfluidic channel with periodically arranged diagonal ridges that separate micrometer-sized particles by size. We use a hybrid numerical method that combines the lattice Boltzmann model (LBM) and lattice spring model (LSM) to examine the dynamics of suspended particles in such channels. Our simulations reveal that particles with different sizes follow distinct trajectories and separate in the lateral direction inside ridged microchannels. The trajectories are determined by the particle equilibrium position in narrow constrictions formed diagonal ridges. We characterize the separation performance by analyzing the effects of ridge geometry and compare our simulation results with experimental data. This microfluidic system can be employed for high throughput sorting and separation of biological cells and synthetic microparticles.

Support from NSF (CBET-0932510) is gratefully acknowledged.

9:05AM GP.00006 Continuous Electrokinetic Separation of Particles and Cells in a Serpentine Microchannel, CAMERON CANTER, JUNJIE ZHU, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921, USA, TZUEN RONG TZENG, Department of Biological Sciences, Clemson University, Clemson, SC 29634-0314, XIANCHUN XUAN, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921, USA — Particle separation plays an important role in many areas. A variety of force fields have been used to separate particles in microfluidic devices, among which electric field may be the most popular one due to its general applicability and adaptability. So far, however, electrophoresis-based separations have been limited to primarily batchwise processes. Dielectrophoresis (DEP)-based separations require in-channel micro-electrodes or micro-insulators to produce electric field gradients. In this talk we present a novel electrokinetic separation of particles in a serpentine microchannel. The continuous separation arises from the cross-stream particle dielectrophoresis induced by the non-uniform electric field inherent to curving microchannels. We demonstrate a size-based separation of polystyrene beads (1 \( \mu \)m and 5 \( \mu \)m in diameter) and microbial cells (E. coli and yeast) in the serpentine microchannel with the application of a small DC electric field. We also develop a numerical model to simulate the particle and cell separation processes.

9:18AM GP.00007 Electrokinetic Transport and Manipulation of Particles in Microfluidic Reservoirs, JUNJIE ZHU, XIANCHUN XUAN, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921 — Electrically controlled microfluidic devices have been proven to be very useful in manipulating both synthetic and biological particles in terms of efficiency, sensitivity, and simplicity. The success of these devices depends on a comprehensive understanding of electrokinetic particle transport in every part of their microchannels and reservoirs. In this talk we present an experimental and numerical study of the electrokinetic transport of spherical polystyrene beads in microfluidic reservoirs. We also demonstrate that polystyrene beads can be continuously focused, trapped, concentrated, and separated in microfluidic reservoirs. This diverse electrical control of particle transport in reservoirs is envisioned to open new possibilities for handling bioparticles in electrokinetic microfluidic systems.
Optoelectrofluidic field separation based on light-intensity gradients and its applications

1 JIN-SUNG YOON, SANG-HYUN LEE, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — Optoelectrofluidic field separation (OEF) of particles under light-intensity gradient (LIG) is reported. The LIG illumination on the photoconductive layer converts the short-range dielectrophoresis (DEP) force to the long-ranged one. The long-ranged DEP force can compete with the hydrodynamic force by alternating current electro-osmosis (ACEO) over the entire illumination area for realizing effective field separation of particles. Results of the field separation and concentration of diverse particle pairs (0.82–16 μm) are well demonstrated, and conditions determining the critical radius and effective particle manipulation are discussed. In addition, expanding the OEFs to biological applications such as rapid cell manipulation and separation will be discussed. The OEFs with LIG strategy could be a promising manipulation method of particles including biological cells in many applications where a rapid manipulation of particles over the entire working area is of interest.

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) (Grant No. R0A-2007-000-20098-0).

Deterministic Particle Trapping in Laminar Microvortices

ALBERT MACH, DINO DI CARLO, University of California, Los Angeles — We present a method of deterministic trapping of particles larger than a critical size in laminar microscale vortices. This novel phenomenon is observed in microchannels containing a straight channel with periodic expansion and contraction arrays. High fluid flow rates in the laminar regime create a detached boundary layer in each array producing two symmetric fluid recirculation zones. Particles introduced into the straight channel experience a two-layer lift for, due to shear gradient and wall effect when inertia is important. As particles approach the expansion, larger shear gradient lift induces larger particles to migrate laterally across streamlines and into the vortex, since the balancing wall-effect lift is no longer significant immediately after the expansion. Smaller particles are maintained in streamlines that flow out of the device because they experience less shear gradient lift—scaling with particle diameter cubed. We identify the hydrodynamic forces responsible for the trapping mechanism, determine the critical particle size for trapping, and present potential biological applications in concentrating cells from complex samples using this phenomenon.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GQ Bubbles I Long Beach Convention Center 203B

Interference between vibrational modes in bubble break-up

S. OBERDICK1, LIPENG LAI, WENDY W. ZHANG, University of Chicago — Recent works reveal that the dynamics near the break-up of an underwater bubble does not evolve into a singular, universal form independent of initial conditions. Instead, any initial azimuthal distortion excites vibrations in the neck shape that dominate the final break-up. Here we investigate how the final break-up is affected by the presence of several different vibrational modes. Approximating the Hamiltonian evolution of the interface as integrable by treating the amplitudes and the phases of the vibrations as action-angle variables gives a simple model of the break-up dynamics. We find that the outcomes of the model are in reasonable agreement with simulation results for most initial distortions. The cross-section of the bubble neck shrinks radially while vibrating. The first break-up occurs when two opposing sides of the interface osculate, creating a smooth contact. One consequence of this vibration-induced break-up is that there exists narrow intervals of initial distortions that evolve into “near-miss” events. In such an event, the two sides of the vibrating interface nearly osculate but pull back just in time. For such initial conditions, the simulated evolution deviates significantly from the model prediction. The action-angle variable approximation also fails.

Current Affiliation: Carnegie Mellon University

Shape instability of a collapsing bubble

ARPIT TIWARI, CARLOS PANTANO, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — A low pressure gas or vapor spherical bubble becomes aspherical during the final stages of the collapse owing to its inherent dynamical instability. We study the nonlinear dynamics of compressible bubble collapses simulated with a three-dimensional HLLC based Riemann solver on an adaptively refined Cartesian mesh. A new interface capturing algorithm is used to preserve the integrity of the Eulerian representation of the gas-liquid interface. The gas is air, which is assumed to be ideal, and the surrounding liquid is water, which is modeled by a stiffened equation of state. Departure from the spherically symmetric Gilmore/Keller–Miksis model is quantified via spherical harmonic spectra of the surface shape. Broadening, and redistribution, of the initial modes as a function of time is observed during the collapse. Simulation results indicate that unexpected care is required to avoid spurious excitation of modes by the far-field boundaries of the computational domain.

Optoelectrofluidic field separation based on light-intensity gradients and its applications

8:00AM GQ.00001 Interference between vibrational modes in bubble break-up

8:13AM GQ.00002 Shape instability of a collapsing bubble

8:26AM GQ.00003 Stability of surface nanobubbles

8:31AM GQ.00004 Numerical simulations of single-bubble collapse in liquid metal

University of Michigan, Ann Arbor — Bubble collapse following a thermal shock in liquid mercury is investigated to understand the resulting cavitation erosion. A conservative high-order accurate interface- and shock-capturing scheme is used to carry out direct simulations of the three-dimensional collapse of a single bubble. Both the shock-induced collapse due to the propagating shock and acoustic waves and the inertial Rayleigh collapse of a cavitation bubble are studied in stationary and laminar flow configurations near a rigid wall. The non-spherical collapse and emitted shock waves are characterized for the given configurations. The stresses measured along the solid surface provide indications of the potential damage of bubble collapse and are related to the erosion patterns observed experimentally in the Spallation Neutron Source at Oak Ridge National Laboratory.

This work is partially supported by Oak Ridge Associated Universities.
respectively, the gas and liquid flow rates. Our theory, which is in good agreement with experiments, predicts that bubble size can be expressed as

\[ \sigma_m \propto \left( \frac{Q_g}{Q_l} \right)^{5/12} \]

where \( \sigma_m \) is the membrane tension, \( Q_g \) and \( Q_l \) are the gas and liquid flow rates, respectively. This equation is derived in order to take into account the presence of (one or more) nearby spherical bubbles as well as liquid compressibility effect on the bubble interface.

The method converges to previously models in two distinct limits. First, it reproduces the bubble radius evolution and pressure disturbances induced by a single bubble subjected to a given far field pressure, irrespective of the relative size of the bubble compared to the grid size. Second, it converges to continuum models based on ensemble-averaged equations when there are many bubbles in a cell. The main advantage of the model is that it allows to access to the instantaneous pressure profiles in the liquid rather than the averaged behaviour. The local pressures generated and scattered by bubble dynamics is important for predicting the peak pressures that can be locally achieved in some points of the liquid when violent bubble collapses are encountered.

8:52AM GQ.00005 Surface stability of an encapsulated bubble subjected to an ultrasonic pressure wave. YUNQIAO LIU, KAZUYASU SUGIYAMA, SHU TAKAGI, YOICHIRO MATSUMOTO, Department of Mechanical Engineering, University of Tokyo — A theoretical study on the shape stability of a nearly spherical bubble encapsulated by a hyperelastic membrane in an ultrasonic field is performed. To describe the dynamic balance on the bubble surface, the membrane effects of the in-plane stress and the bending moment are incorporated into the equation set for the perturbed spherical flow of viscous incompressible fluid (Prosperetti, 1977). The spherical motion of the bubble is numerically obtained by solving the Rayleigh-Plesset equation with the elastic stress. The deflection therefrom is linearized and expanded with respect to the Legendre polynomial. Two amplitudes for each shape mode are introduced since the membrane has mobilities not only in the radial direction but also in the tangential direction. The eigenvalue analysis on the system determines the higher-order natural frequency. The derived system is applied to the temporal evolution of the higher-order shape mode. Stability diagrams for the higher-order shape mode are mapped out in the driving amplitude versus driving frequency phase space for various elastic moduli of the membrane. The most unstable driving frequency is found to be approximately integer multiples of the higher-order natural frequency.

9:05AM GQ.00006 Lattice Boltzmann Simulations for High Density Ratio Flows of Multiphase Fluids. YIKUN WEI, YUEHONG QIAN, Institute of Applied Math and Mechanics, Shanghai University — In the present communication, we will show that the compression effect of the Redlich-Kwong equation of state(EOS) is lower than that of the van der Waals (vdW) EOS. The Redlich-Kwong equation of state has a better agreement with experimental data for the coexistence curve than the van der Waals (vdW) EOS. We implement the Redlich-Kwong EOS in the lattice Boltzmann simulations via a pseudo-potential. As a result, multi-phase flows with large density ratios may be simulated, thus many real applications in engineering problems can be applied. Acknowledgement: This research is supported in part by Ministry of Education in China via project IRT0844 and NSFC project 10625210 and Shanghai Sci and Tech. Com. Project 08Z2Z43

9:18AM GQ.00007 Microjet formation in a capillary by laser-induced cavitation. IVO R. PETERS, YOSHISHUKI TAGAWA, DEVARAJ VAN DER MEER, ANDREA PROSPERETTI, CHAO SUN, DETLEF LOHSE, University of Twente, The Netherlands — A vapor bubble is created by focusing a laser pulse inside a capillary that is partially filled with water. Upon creation of the bubble, a shock wave travels through the capillary. When this shock wave meets the meniscus of the air-water interface, a thin jet is created that travels at very high speeds. A crucial ingredient for the creation of the jet is the shape of the meniscus, which is responsible for focusing the energy provided by the shock wave. We examine the formation of this jet numerically using a boundary integral method, where we prepare an initial interface at rest inside a tube with a diameter ranging from 50 to 500 µm. To simulate the effect of the bubble we then apply a short, strong pressure pulse, after which the jet forms. We investigate the influence of the shape of the meniscus, and pressure amplitude and duration on the jet formation. The jet shape and velocity obtained by the simulation compare well with experimental data, and provides good insight in the origin of the jet.

9:31AM GQ.00008 Cavitation damage in blood clots under HIFU. HOPE WEISS, Department of Mechanical Engineering, University of California, Berkeley, GOLNAZ AHADI, THILO HOELSCHER, Department of Radiology, University of California, San Diego, ANDREW SZERI, Department of Mechanical Engineering, University of California, Berkeley — High Intensity Focused Ultrasound (HIFU) has been shown to accelerate thrombolysis, the dissolution of blood clots, in vitro and in vivo, for treatment of ischemic stroke. Cavitation in sonothrombolysis is thought to play an important role, although the mechanisms are not fully understood. The damage to a blood clot associated with bubble collapses in a HIFU field is studied. The region of damage caused by a bubble collapse on the fibrin network of the clot exposed to HIFU is estimated, and compared with experimental assessment of the damage. The mechanical damage to the network caused by a bubble is probed using two independent approaches, a strain based method and an energy based method. Immunofluorescent fibrin staining is used to assess the region of damage experimentally.

9:44AM GQ.00009 Modeling bubble clusters in compressible liquids. DANIEL FUSTER, TIM COLONIUS, California Institute of Technology — We present a new model to simulate the behaviour of bubble clouds in compressible liquids. The method uses a volume-averaged approach and defines the pressure and void fraction relative to a computational cell. Inside the cell, a generalisation of the Keller-Miksis equation is derived in order to take into account the presence of (one or more) nearby spherical bubbles as well as liquid compressibility effect on the bubble interface motion. The method converges to previous models in two distinct limits. First, it reproduces the bubble radius evolution and pressure disturbances induced by a single bubble subjected to a given far field pressure, irrespective of the relative size of the bubble compared to the grid size. Second, it converges to continuum models based on ensemble-averaged equations when there are many bubbles in a cell. The main advantage of the model is that it allows to access to the instantaneous pressure profiles in the liquid rather than the averaged behaviour. The local pressures generated and scattered by bubble dynamics is important for predicting the peak pressures that can be locally achieved in some points of the liquid when violent bubble collapses are encountered.

9:57AM GQ.00010 Modeling bubbles and dissolved gases after a breaking-wave. JUNHONG LIANG, JAMES MCWILLIAMS, UCLA, PETER SULLIVAN, NCAR, BURKARD BASCHEK, UCLA — We developed a bubble concentration model and a dissolved gas concentration model for the oceanic boundary layer. The bubble model solves a set of concentration equations for multiple gases in bubbles of different sizes, and the dissolved gas concentration model simulates the evolution of dissolved gases and dissolved inorganic carbon. The models include the effects of advection, diffusion, bubble buoyant rising, bubble size changes, gas exchange between bubbles and ambient water, and chemical reactions associated with the dissolved CO2. To study the bubble and dissolved gas evolution after a single wave-breaking event, the model is coupled with a fluid-dynamical Direct Numerical Simulation model with spatially and temporally distributed momentum and bubble injection for a typical breaking wave. The modeled bubble size spectrum compares well with the laboratory measurements. The bubble-concentration vortex not only advects the bubble-induced dissolved gas anomalies downstream, but also entrains the surface diffusion layer to greater depth. Due to the hydrostatic pressure and surface tension exerted on bubbles, bubbles do not contribute to the total air-sea gas flux when the water is at a saturation level \( \sigma_m > \sigma_m^b \). When the actual saturation level \( \sigma_m < \sigma_m^b \), the integrated bubble contribution to gas flux is dissolution. When \( \sigma_m > \sigma_m^b \), bubbles add to the venting of dissolved gases.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GR Drops VI Long Beach Convention Center 203C

8:00AM GR.00001 Generation of micron-sized bubbles at the entrance region of PDMS microchannels. JOSE MANUEL GORDILLO, Grupo de Mecanica de Fluidos, Universidad de Sevilla, Spain, WIM VAN HOEVE, DETLEF LOHSE, Physics of Fluids, Twente University, The Netherlands, ELENA DE CASTRO-HERNANDEZ, Grupo de Mecanica de Fluidos, Universidad de Sevilla, Spain — Here we present a new regime of operation of multiphase-based flow focusing microfluidic devices. We show that bubbles with diameters below one tenth the channel width, which we fix here to \( w = 50 \mu m \), can be produced in low viscosity liquids thanks to the strong pressure gradient existing at the entrance region of the channel. Our theory, which is in good agreement with experiments, predicts that bubble size can be expressed as \( d_b/w \propto (Q_2/Q_1)^{5/12} \), where \( Q_2 \) and \( Q_1 \) indicate, respectively, the gas and liquid flow rates.
Exploring the effect of liquid crystalline phase on droplet breakup

ITAI COHEN, JOHN SAVAGE, DAN PORTER, Cornell University, MARCO CAGGIONI, PATRICK SPICER, Procter and Gamble — We investigate droplet breakup of a thermotropic liquid crystal in the smectic, nematic, and isotropic phases. The experiment consists of varying the ambient temperature to control the liquid crystalline phase and imaging breakup using a fast video camera. We find breakup of the smectic phase is well described by existing theory for a shear thinning power-law fluid. These theories predict the stress/strain dependence measured in bulk rheology coincides with the radial radius dependence on time to breakup. For nematic liquid crystals, a new breakup model is developed based on new measurements that indicate Newtonian behavior. Instead, breakup occurs in two stages, with extensional thickening preceding extensional thinning. Finally, we will comment that the bulk concentration is greater than the critical micelle concentration. We use a one-dimensional kinetic-limited transport model to demonstrate that local surface coverages can lead to highly nonlinear effects like tipstreaming at these length and time scales.

A model for predicting drop size distribution in effervescent jet breakup

This project was supported by the EPSRC and industrial partners in the Innovation in Inkjet Technology and Innovation in Industrial Inkjet Technology projects.

A method to generate picoliter droplets out of a microliter drop, on-demand using satellite formation

DUSTIN MOON, DO JIN IM, IN SEOK KANG, POSTECH — We investigated a simple, robust way to generate pico- to femtoliter drops out of a single microliter droplet for the use of generating monodisperse droplets in droplet-based microfluidics. A single droplet is placed between glass substrates, immersed in silicone oil with different viscosities, moved with constant velocities from 50micron/s to 1500micron/s. As two plates separates, liquid bridge breaks and smaller droplets, or satellites are formed. We have found that for a fixed viscosity, nearly same size of droplets are generated over several orders of velocities. Using this single, cell encapsulation is also possible without any other complex control and we successively captured a single Arabidopsis Protoplast with this method. This method can be used to divide small bio sample on-demand, to several smaller droplets for further analysis.

A method for generating droplet breakup out of a thermotropic liquid crystal

The experiment consists of varying the ambient temperature to control the liquid crystalline phase and imaging breakup using a fast video camera. We find breakup of the smectic phase is well described by existing theory for a shear thinning power-law fluid. These theories predict the stress/strain dependence measured in bulk rheology coincides with the radial radius dependence on time to breakup. For nematic liquid crystals, a new breakup model is developed based on new measurements that indicate Newtonian behavior. Instead, breakup occurs in two stages, with extensional thickening preceding extensional thinning. Finally, we will comment that the bulk concentration is greater than the critical micelle concentration. We use a one-dimensional kinetic-limited transport model to demonstrate that local surface coverages can lead to highly nonlinear effects like tipstreaming at these length and time scales.

A method to predict the effects of liquid crystalline phase on droplet breakup

This project was supported by the EPSRC and industrial partners in the Innovation in Inkjet Technology and Innovation in Industrial Inkjet Technology projects.
9:44AM GR.00009 Breakup of double emulsions in wedge-shaped microfluidic channels\textsuperscript{1}, \textsuperscript{2} JIANG LI\textsuperscript{2}, University of Science and Technology Beijing, HAOSHENG CHEN\textsuperscript{2}, Tsinghua University, HOWARD A. STONE, Princeton University — Double emulsion droplets can serve as drug delivery vehicles and individual compartments for chemical reactions, and such materials are relevant to new kinds of microfluidic applications. We study experimentally the dynamics and breakup of double emulsion droplets flowing through poly(dimethylsiloxane) (PDMS) channels. As water-in-oil-in-water (W/O/W) double emulsion droplets flow through such wedge-shaped channels, the breakup of the droplets is controlled by the capillary number and the droplet-to-orifice size ratio. We obtain a phase diagram of droplet breakup morphology from the experimental results, and explain the results via a combination of the capillary instability and thin film dynamics. The phase diagram is useful for predicting and controlling the breakup of the droplet. Finally, differences between results obtained in PDMS channels and capillary channels are discussed.

\textsuperscript{1}Supported by NSFC Projects (No. 50805008 and 50975158) and Fundamental Research Funds for Central Universities (FRF-TP-09-013A)
\textsuperscript{2}Current visiting Princeton University
\textsuperscript{3}Current visiting Harvard University

9:57AM GR.00010 Evolution, Pinch-off, and Classification of the Steady-State Solutions of an Axisymmetric Drop in Stokes Flow, SHADI NADERI, MONIKA NITSCHER, University of New Mexico — The evolution of an axisymmetric viscous drop immersed in a strain field is examined numerically using higher order boundary integral simulation followed the work of Nitsche et al. [J. Comp. 229, 2010]. The effect of three parameters is examined, namely: the capillary number, the viscosity ratio and the relative nonlinearity in background flow. A classification of the steady-state solutions in parameter space for sufficiently small capillary number is presented including the regimes of oval, canonical and bell-shaped steady-states. The non-steady evolution for larger capillary number is also studied and classified. New results include the effect of the nonlinearity in background flow. The presence of a positive nonlinear term leads to corner formation as time goes to infinity. Negative nonlinearity on the other hand leads to a finite time singularity. Either the drop pinches at two points on the axis in finite time or the curvature blows up at a point away from the axis. Pinch-off is also investigated in detail.

Monday, November 22, 2010 8:00AM - 10:10AM – Session GS Drops VII Long Beach Convention Center Grand Ballroom A

8:00AM GS.00001 Effect of neighboring particles on drop coalescence at an interface\textsuperscript{1}, ANKUR BORDOLOI, DEEPAK ADHIKARI, ELLEN LONGMIRE, Dept of Aerospace Engineering and Mechanics, University of Minnesota — The coalescence of a liquid drop in the presence of an adjacent solid particle or liquid drop is studied using high-speed visualization and Tomographic PIV. A drop of water/glycerin (W/G), surrounded by silicone oil of matched refractive index, is released onto an underlying W/G interface. A nylon sphere, neutrally buoyant with respect to the drop liquid, is placed adjacent to the drop. Three initial conditions are considered: the particle is wetted in W/G so that the interface maintains an angle of contact with the particle, the particle is wetted in oil so that it rests above the interface, and the particle is placed so that it maintains an angle of contact with the drop already resting above the interface. These cases are compared with that of two neighboring W/G drops. Off-axis rupture near the solid particle was found to be dominant in cases where the particle was wetted with W/G. However, when the particle was wetted with oil, the point of rupture occurred closer to the drop-axis. The film rupture in the drop is followed by retraction of the film and finally collapse of the drop. Both visualization and PIV results show that the trajectory of the collapsing drop depends on the initial contact condition as well as the rupture location.

\textsuperscript{1}Supported by DOE EERE-PMC-10EE0002764.

8:13AM GS.00002 Coalescent coalescence , NICOLAS BREMOND, HUGO DOMEJEAN, JÉRÔME BIBETTE, ESPCI ParisTech/LCMD — Pulling apart two neighboring emulsion drops favors their coalescence. This counterintuitive phenomenon is then responsible for the propagation of coalescence among a concentrated emulsion. Indeed, the shape relaxation during the coalescence phase can induce a separation with the neighboring drops, a situation that is potentially destabilizing. We report an experimental investigation on such catastrophic phenomenon by using a microfluidic device where a calibrated two-dimensional emulsion is created and destabilized. The velocity of propagation as well as the probability of the coalescence are reported as a function of the size and the spatial distribution of the drops. We then propose a scenario for phase inversion of a concentrated emulsion based on this mechanism and discuss its efficiency by taking into account the existence of a drop size distribution.

8:26AM GS.00003 Viscous to Inertial Crossover in Liquid Drop Coalescence , JOSEPH PAULSEN, JUSTIN BURTON, SIDNEY NAGEL, James Franck Institute, University of Chicago — When two liquid drops coalesce, a dramatic topological transition occurs. We study experimentally the dynamics and breakup of double emulsion droplets flowing through poly(dimethylsiloxane) (PDMS) channels. As water-in-oil-in-water (W/O/W) double emulsion droplets flow through such wedge-shaped channels, the breakup of the droplets is controlled by the capillary number and the droplet-to-orifice size ratio. We obtain a phase diagram of droplet breakup morphology from the experimental results, and explain the results via a combination of the capillary instability and thin film dynamics. The phase diagram is useful for predicting and controlling the breakup of the droplet. Finally, differences between results obtained in PDMS channels and capillary channels are discussed.

8:39AM GS.00004 Mass-spring-damper dynamic system modeling for predicting drop-pair interaction outcomes , PAUL VAN NOORDT, MICAH BERGMAN, CARLOS HIDROVO, The University of Texas at Austin — In the present study, we investigate both theoretically and experimentally the process of two drops interacting through a head-on collision and the various outcomes that may result. The relationship between kinetic and surface energy of the colliding drop pair, as well as the viscosity of the intervening gaseous medium, are considered as factors that govern the outcome of the collision. A theoretical model is derived, which treats the collision process as a squeeze-film problem involving both planar and non-planar geometries. Based on the various mechanisms that influence the collision dynamics, an analogy is made between the fluidic system of liquid drops and a mechanical mass-spring-damper system. Examination of the analogous mechanical system yields an equivalent damping ratio, which is used to predict the outcome of the drop-pair collision. Our experimental setup allows drops of varying speed and size to interact with each other in a mid flight collision. The collision process is captured using high-speed photography, and the results obtained are used to validate our theoretical model and the effectiveness of our damping ratio in predicting the outcome of drop-pair collisions.
Boltzmann method, KRZYSZTOF KUBIAK, University of Leeds, M.C.T. WILSON, J.L. SUMMERS, N. KAPUR, K. HOOD — Droplet coalescence on advective Cahn-Hilliard formulations. [Phys. Fluids 20, 046602 (2008)], are used to study the effects of small concentrations of insoluble surfactants C20, droplet coalescence can be properly simulated and the final footprint can be predicted.

The lattice Boltzmann method presents great potential for coalescence modelling especially on non-uniform surfaces. The dynamics of data. LBM simulations in 3D allow a comparison of the final footprint of droplet. Introduction of a non-uniform surface wettability into the model, in the form of hydrophilic patterns, helps to obtain final footprints of ellipsoidal form. Different size and distribution of patterns has been studied to analyze its influence on the coalescence process. The lattice Boltzmann method presents great potential for coalescence modelling especially on non-uniform surfaces. The dynamics of droplet coalescence can be properly simulated and the final footprint can be predicted.

9:05AM GS.00006 Effects of small concentration surfactants on the coalescence of viscous drops1. CAROLINA VANNONZO, University of California Santa Barbara — Boundary integral simulations, employing Dai and Leal’s code [Phys. Fluids 20, 046602 (2008)], are used to study the effects of small concentrations of insoluble surfactants C, on head-on collisions of two equal-sized viscous drops in a matrix of equal viscosity in a hyperbolic extensional flow, for low Reynolds numbers. The parameters were chosen to mimic the experiments of Yoon et al. [Phys. Fluids 19, 023102 (2007)], which were performed with polymeric drops stabilized by block-copolymer insoluble surfactants in a polymer matrix, where both fluids acted as Newtonian viscous fluids. In these experiments a discontinuous transition in the coalescence process was found for low C as the Capillary number Ca was increased. Thus, for Ca ≈ Ca, a minimum surfactant concentration exists below which the system behaves like a clean interface system. Here, by varying C, i.e. the Marangoni number Ma, and the surface diffusivity, i.e. the interfacial Peclet number Pe, we explain the origin of the transition and its dependence on the parameters. The transition occurs if Pe > Pe, Ca > Ca, and Ma > Ma.

1 This work was partially supported by NSF grant 0624446

9:18AM GS.00007 Analysis of the formation of drops of a Bingham fluid1. HAILING GAO, SANTOSH APPATHURAI, PATRICK MCGOUGH, MICHAEL HARRISS, OSMAN BASARAN, Purdue University — Emulsions, dispersions, and foams are both of scientific interest and widely used in technological applications. A way to form such dispersed systems is to flow a liquid or a gas from a tube into a continuous phase of another fluid. In this talk, the dynamics of formation of drops of a Bingham fluid from a tube into a gas are studied computationally. The dynamics are governed by four dimensionless groups: Ohnesorge number, Oh (dimensionless drop viscosity), Weber number, We (square root of dimensionless flow rate), Bond number, G (ratio of gravitational to surface tension force), and yield stress parameter, Y (ratio of yield stress to capillary pressure). Tracking the evolution in time of yielded and unyielded regions in the drop is shown to be crucial for developing a good understanding of the fluid dynamics of the process. The effects of the governing dimensionless groups on the volumes of the primary drops that are formed and whether small satellites as well as large primary drops are produced are investigated. Similarities and differences between the dynamics of formation of drops of Bingham fluids and those of Newtonian fluids are also elucidated.

1 Research supported by Chevron and P&G.

9:31AM GS.00008 Analysis of the formation of drops of a Herschel-Bulkley fluid. PATRICK MCGOUGH, SANTOSH APPATHURAI, HAILING GAO, OSMAN BASARAN, Purdue University — Although viscoplastic liquids are widely used in technological applications, study of dynamics of drops of such liquids has received little attention to date. In this talk, the dynamics of formation of drops of a Herschel-Bulkley fluid from a tube into a gas are studied computationally and experimentally. The dynamics are governed by five dimensionless groups: Ohnesorge number, Oh (dimensionless drop viscosity), Weber number, We (square root of dimensionless flow rate), Bond number, G (ratio of gravitational to surface tension force), power-law exponent, n, and yield stress parameter, Y (ratio of yield stress to capillary pressure). Computational results are matched against experimental results. Tracking the evolution (yielded and unyielded) of regions in the drop is shown to be crucial for developing a good understanding of the fluid dynamics of the process. The effects of the governing dimensionless groups on the volumes of the primary drops that are formed and whether small satellites as well as large primary drops are produced are investigated. Similarities and differences between the dynamics of formation of drops of Herschel-Bulkley fluids and those of Newtonian fluids are also elucidated.

9:44AM GS.00009 Probing Interfacial Emulsion Stability Controls using Electrorheology1. XIUYU WANG, AMY BRANDVIK, VLADIMIR ALVARADO, University of Wyoming — The stability of water-in-oil emulsions is controlled by interfacial mechanisms that include oil film rheology of approaching drops and the strength of drop interfaces. Film drainage is mainly a function of the continuous phase rheology. Temperature is used to regulate the viscosity of the continuous phase and hence determine its effect on emulsion stability through film drainage, in contrast with interfacial strength. In this study, one crude oil is used to formulate water-in-oil emulsions. Oil-water interfacial tension is measured to gauge other interfacial changes with temperature. The critical field value, used as proxy of emulsion stability, approaches a plateau value for each crude oil- aqueous solution pair, which is interpreted to reflect the intrinsic drop-coating film resistance to coalescence. Interfacial tension does vary significantly with either aqueous phase composition or temperature. From comparison with previous results, we speculate that drop coating film is composed of a fraction of asphaltic compunds.

1 This material is based upon work supported by the University of Wyoming School of Energy Resources through its Graduate Assistantship program and the Enhanced Oil Recovery Institute

9:57AM GS.00010 Stirring a Cahn-Hilliard fluid in moving microdroplets, SAIF A. KHAN, S.H. SOPHIA LEE, PENGZHI WANG, SWEE KUN YAP, National University of Singapore — Biochemistry within living eukaryotic cells occurs in dynamic heterogeneous fluid environments containing macromolecules such as proteins, nucleic acids and sugars; most in-vitro biochemical studies in dilute aqueous solutions do not capture this chemical and morphological complexity. Here, as an in-vitro model for in-vivo cellular environments, we investigate the dynamics of a phase separating aqueous polymer mixture in small moving droplets. We dispense aqueous mixtures of poly(ethylene glycol) (PEG) and dextran as droplets carried by an immiscible fluorinated oil at a microfluidic T-junction, and use high-speed optical microscopic imaging to observe dynamic phase behavior. In the static case, for off-critical compositions, this mixture separates via a spinodal mechanism into two phases- a PEG-rich phase and a dextran-rich phase. For moving drops, the polymer mixture exhibits a near continuum of flow and composition-dependent phase morphologies, from the ‘unmixed’ static morphology to complex percolated morphologies resembling in vivo cellular environments. We compare our measurements to previous theoretical and numerical studies of binary fluid mixing based on advective Cahn-Hilliard formulations.
8:00AM GT.00001 Falling flexible sheets, SILAS ALBEN, Georgia Tech — We use inviscid simulations to study falling flexible sheets in the two-parameter space of sheet density and bending rigidity. The basic behavior is a repeated series of accelerations to a critical speed at which the sheet flexes, and rapidly decelerates, shedding large vortices. The maximum and average speeds of the sheet are closely related to the critical flutter speed. The sheet trajectories also show persistent circling, quasi-periodic flapping, and more complex repeated patterns.

8:13AM GT.00002 Fluttering dynamics of passive flexible wings, DANIEL TAM, JOHN W.M. BUSH, MIT, PHYSICAL MATHEMATICS TEAM — We investigate the dynamics of passive flexible wings freely falling under the influence of gravity. Particular attention is given to elucidating the role of flexibility in gliding flight, specifically side-to-side fluttering motion. The effect of bending on the dynamics of fluttering wings is examined through an experimental investigation of deformable rectangular wings falling in water. We demonstrate that the elastic deformations induced by the flow strongly affect the flight characteristics, specifically the period and amplitude of the side to side fluttering motion as well as the descent rate. Our results suggest the existence of an optimal bending rigidity that maximizes the descent time for a particular wing geometry. Biological implications are discussed.

8:26AM GT.00003 Falling with Style - Bat flight maneuvers, ATTILA BERGOU, Brown University, DANIEL RISKIN, City College New York, GABRIEL TAUBIN, SHARON SWARTZ, KENNETH S. BREUER, Brown University — The remarkable maneuverability of flying animals results from precise movements of their highly specialized wings. Among these flyers, bats have evolved a particularly impressive capacity to control their flight. This adeptness is, in part, determined by bats’ ability to modulate their wing shape through many independently controlled joints. However, the many-jointed wings of bats have higher inertia relative to their bodies compared with all other extant flyers. To understand the role that wing inertia plays in bat flight, we use a novel tracking algorithm to measure the kinematics of bats performing aerial flips. Using a dynamical model of a flying bat, we show how bats modulate their wings’ inertia, usually a detriment to maneuvering, to supplement aerodynamic forces in performing flight maneuvers.

8:39AM GT.00004 Effect of mass ratio for a flexible flapping wing during forward flight1, HAOXIANG LUO, Vanderbilt University, FANG-BAO TIAN, XI-YUN LU, University of Science and Technology of China — During flight, insect wings typically deform under a combined aerodynamic force and wing inertia, whichever is dominant depends on the properly scaled mass ratio between the wing and air. To study the differences that the wing inertia makes in the aerodynamic performance of the deformable wing, a two- dimensional numerical study is applied to simulate the flow– structure interaction of a flapping wing during forward flight. The wing section is modeled as an elastic plate that may experience nonlinear deformations while flapping. The effect of the wing inertia on lift, thrust, and power is studied for a range of wing rigidity and kinematic parameters such as the stroke plane angle and advance angle. It is found that the wing flexibility can dramatically increase the thrust without significantly losing lift or increasing the power input. Furthermore, the wings with low mass ratios could have much better efficiency than the wings with high mass ratios. The implication of the findings on insect flight will be discussed.

8:52AM GT.00005 Thrust performance and wake structure of a pitching flexible plate at low aspect ratio1, PAULO FERREIRA DE SOUSA, HU DAI, HAOXIANG LUO, Vanderbilt University, JAMES DOYLE, Purdue University — Thrust performance and wake structure are numerically investigated for a rectangular plate (AR = 0.54) that pitches around the leading edge in a free stream. The plate is flexible and it may undergo large displacement. The simulations employ a newly developed fluid-structure-interaction code based on a sharp-interface immersed boundary solver for the flow and a nonlinear finite-element solver for the elastic plate. Implemented on a Cartesian mesh, the flow solver allows us to capture the vortex dynamics of the wake accurately and efficiently. The mass ratio of the plate is low so that the deformation is solely caused by the hydrodynamic force. The results will be compared with the experimental result for the rigid plate from Buchholz and Smits (J Fluid Mech 603, 2008). Both the thrust level and power efficiency will be used to evaluate the performance of the plate, and the results will be compared with those for the corresponding rigid plate with the same effective pitching angle. The effect of the active pitching angle, the bending rigidity, and the Strouhal number will be presented.

9:05AM GT.00006 Efficient flapping flight using flexible wings oscillating at resonance1, ALEXANDER ALEXEEV, HASSAN MASOUD, Georgia Institute of Technology — Using a fully-coupled computational approach that integrates the lattice Boltzmann and lattice spring models, we investigate the three-dimensional aerodynamics of flexible flapping wings at resonance. The wings are tilted from the horizontal and oscillate vertically driven by a force applied at the wing root. Our simulations reveal that resonance oscillations drastically enhance the aerodynamic efficiency of low-Reynolds-number plunging, and yield lift and lift-to-weight ratio comparable to the values typical for small insects. Within the resonance band, we identify two flapping regimes leading to the maximum lift and the maximum efficiency, which are characterized by different bending modes of flexible flapping wings. Our results indicate the feasibility of using flexible wings driven by a simple harmonic stroke for designing efficient microscale flying machines.

9:18AM GT.00007 On passive wing response in hovering kinematics at low Reynolds number, ALBERT MEDINA, JEFF D. ELDREDGE, Mechanical & Aerospace Engineering, University of California, Los Angeles, CA, USA — The aerodynamic role of passive wing mechanics in biological flight poorly understood. This computational study focuses on the effects of passive pitch response via chordwise flexibility in two-dimensional hovering kinematics. The wing consists of solid elliptical bodies with interconnected torsional springs. Tests of a wing subjected to a nominally fixed pitch angle reveal new mechanisms for mean lift generation through symmetry-breaking. Additionally, chordwise flexibility in hovering kinematics accounts for greater average lift production and efficiency than the equivalent rigid wing. It is shown that an effective angle of attack can be defined that collapses the performance of the flexible wing to that of a rigid wing. Further, it is found that the performance of a flexible wing undergoing hovering maneuvers is less sensitive to pitch amplitude and phase. Finally, these results are put into context with recent studies of passive pitching of a rigid wing, and a more general class of passive wing behaviors is identified.

9:31AM GT.00008 The effect of chordwise flexibility on flapping foil propulsion in quiescent fluid, SACHIN SHINDE, JAYVANT ARAKERI, Indian Institute of Science, Bangalore-12 — Motivated to understand the role of wing flexibility of flying creatures during hovering, we experimentally study the effect of chordwise flexibility on the flow generated in quiescent fluid by a sinusoidally pitching rigid symmetrical foil with a flexible flap attached at the trailing edge. This foil produces a narrow, coherent jet containing reverse Karman vortex street, and a corresponding thrust. The thrust and flow is similar to that produced by a hovering bird or insect, however the mechanism seems to be different from known hovering mechanisms. Novelty of the present hovering mechanism is that the thrust generation is due to the coordinated pushing action of rigid foil and flexible flap. We identify the flow and vortex generation mechanism. This foil produces jet flows over a range of flapping frequencies and amplitudes. In contrast, the foil without flap i.e. with rigid trailing edge produces a weak, divergent jet that meanders randomly. Appending a flexible flap to the foil suppresses jet-meandering and strengthens the jet. Flexibility of flap is vital in determining the flow structure. This study is useful in designing MAVs and thrusters.
9:44AM GT.00009 Flexible Flapping Foils\(^1\), CATHERINE MARAIS, RAMIRO GODOY-DIANA, JOSÉ EDUARDO WESFREID, PMMH UMR7636 CNRS; ESPCI ParisTech; UPMC, Université Paris Diderot — Hydrodynamic tunnel experiments with flexible flapping foils of 4:1 span-to-chord aspect ratio are used in the present work to study the effect of foil compliance in the dynamical features of a propulsive wake. The average thrust force produced by the foil is estimated from 2D PIV measurements and the regime transitions in the wake are characterized according to a fluctuating frequency-amplitude phase diagram as in Godoy-Diana et al. (Phys. Rev. E 77, 016308, 2008). We show that the thrust production regime occurs on a broader region of the parameter space for flexible foils, with propulsive forces up to 3 times greater than for the rigid case. We examine in detail the vortex generation at the trailing edge of the foils, and propose a mechanism to explain how foil deformation leads to an optimization of propulsion.

\(^1\)We acknowledge the French Research Agency for support through Project No. ANR-08- BLAN-0099.

9:57AM GT.00010 Bendable ring flapping in a uniform flow\(^1\), BO YOUNG KIM, SOO JAI SHIN, HYUNG JIN SUNG, KAIST — To understand flow-induced flapping motions of bendable objects, we numerically investigate dynamics of a pressurized elastic ring pinned at one point within a uniform flow by using an improved version of the immersed boundary method. The boundary of the ring consists of a flexible filament with bending stiffness, which can be modeled as a linear spring with spring constant \(k\) and initial unstretched length. The internal area of the ring is conserved through the penalty method. The flapping motion of the ring is decomposed into two parts: a pitching motion that includes flexible bending motion in the transverse direction, and a tapping motion in the longitudinal direction. For the Reynolds number of 100, resonance is observed at \(k \sim 11\), where \(k\) is normalized by the diameter of the undeformed ring, the speed of the upcoming flow and the fluid density. Across the resonance region, an abrupt jump in terms of the motion amplitudes as well as the hydrodynamic loads is recorded. In our simulation we observe bistable states, one stationary and another oscillatory, that coexist over a range of flow velocities.

\(^1\)This work was supported by the Creative Research Initiatives and World Class University programs of MEST/NRF.

Monday, November 22, 2010 8:00AM - 10:10AM — Session GU Multiphase Flows IV — Hyatt Regency Long Beach Regency A

8:00AM GU.00001 Two-phase PIV measurements of particle suspension in a forced impinging jet\(^1\), RAHUL MULINTI, KEN KIGER, University of Maryland — The condition of rotorcraft brownout is characterized by intense dust suspension that is uplifted during landing and takeoff operations in regions covered with loose sediment. To predict particle suspension and sedimentation within coupled particle-laden flows, detailed characterization of the micro-scale mechanics is needed within a prototypical flow that captures the essence of the rotorcraft/ground wake interactions. Two-phase PIV has been used to study the interaction of a sediment bed made of glass spheres with characteristic flow structures reminiscent from flow within a rotor wake. In order to make reliable simultaneous two-phase PIV measurements, a phase discrimination algorithm from a single two-phase image has been implemented. The validity of the separation is checked by processing images that consisted only of the very small tracer particles, or only the dispersed phase particles, and examining how much “cross-talk” was present between the phases. The mobilization and wall-normal flux of particulates by the vortex-wall interaction will be reported for several different operational conditions, and correlated to the local vortex conditions.

\(^1\)The authors would like to acknowledge the support of AFOSR under grant FA9550-08-1-0406.

8:13AM GU.00002 Impingement of a Planar Shock Wave on a Dense Field of Particles, JUSTIN WAGNER, STEVEN BERESH, SEAN KEARNEY, WAYNE TROTT, JAIME CASTANEDA, BRIAN PRUETT, MELVIN BAER, Sandia National Lab — A novel multiphase shock tube has recently been developed to study particle dynamics in gas-solid flows having particle volume fractions that reside between the dilute and granular regimes. The method for introducing particles into the tube involves the use of a gravity-fed contoured particle seeder, which is capable of producing dense fields of spatially isotropic particles. The facility is capable of producing planar shocks having a maximum shock Mach number of about 2.1 that propagate into air at ambient initial conditions. The primary purpose of this new facility is to provide high fidelity data of shock-particle interactions in flows having particle volume fractions of about 1 to 50%. To achieve this goal, the facility drives a planar shock into a spatially isotropic field, or curtain, of particles. Experiments are conducted for two configurations where the particle curtain is either parallel to the spanwise, or the streamwise direction. Arrays of high-frequency-response pressure transducers are placed near the particle curtain to measure the attenuation and shape change of the shock owing to its interaction with the dense gas particle field. In addition, simultaneous high-speed imaging is used to visualize the impact of the shock on the particle curtain and to measure the particle motion induced downstream of the shock.

8:26AM GU.00003 Multiphysics Simulations of Hot-Spot Initiation in Shocked Insensitive High-Explosive\(^1\), FADY NAJJAR, W.M. HOWARD, L.E. FRIED, Lawrence Livermore National Laboratory (LLNL) — Solid plastic-bonded high-explosive materials consist of crystals with micron-sized pores embedded. Under mechanical or thermal insults, these voids increase the ease of shock initiation by generating high-temperature regions during their collapse that might lead to ignition. Understanding the mechanisms of hot-spot initiation has significant research interest due to safety, reliability and development of new insensitive munitions. Multi-dimensional high-resolution meso-scale simulations are performed using the multiphysics software, ALE3D, to understand the hot-spot initiation. The Cheetah code is coupled to ALE3D, creating multi-dimensional sparse tables for the HE properties. The reaction rates were obtained from MD Quantum computations. Our current predictions showcase several interesting features regarding hot-spot dynamics including the formation of a “secondary” jet. We will discuss the results obtained with hydro-thermo-chemical processes leading to ignition growth for various pore sizes and different shock pressures.

\(^1\)LLNL-ABS-446352. This work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

8:39AM GU.00004 A high-resolution numerical method for supercritical flows, HIROSHI TERAISHIMA, Japan Aerospace Exploration Agency, SOSHI KAWAI, Stanford university, NOBUHIRO YAMANISHI, Japan Aerospace Exploration Agency — We present a high-resolution methodology using a higher-order compact differencing scheme with localized artificial diffusivity for simulating cryogenic turbulent mixing flows under supercritical thermodynamic conditions. One-dimensional advection and modified Shu-Osher problems in supercritical flows are proposed to assess the performance of the present method. Results for the advection problem show that the present method is successfully applied to supercritical flows, including a trans-critical state, without any significant spurious oscillations, if initial startup errors are avoided. A localized artificial diffusivity, especially artificial thermal conductivity for temperature gradients, effectively works on reducing numerical wiggles produced due to high density/temperature gradients at interfaces. The modified Shu-Osher problem demonstrates the superiority of the present method in resolving high-frequency fluctuations as compared to a conventional upwind-biased scheme. Results for a two-dimensional cryogenic planar jet in a supercritical pressure condition also demonstrate the capability of the present method for simulating the unsteady jet flow structures and the superiority for resolving the fluctuations with reasonable grid resolutions.
Recent developments of the method have significantly improved its stability and accuracy for this type of calculations. The use of the method to fluid-structure interactions on the interface region. Furthermore, if the solid material is a porous material, material interactions occur not only on the interface region, but also inside the body of the materials. Recently, this type of problems has been studied using a continuous multiphase flow theory. Material point method to fluid-structure interactions has produced many promising results. In this talk we will briefly introduce the recent developments of MPM and shown examples of its applications.

One of the advantages of the Material Point Method (MPM) is its capability to simulate large material deformation and flows without the need to advect state variables, such as stress and strain of the material through an Eulerian mesh. Without numerical diffusion associated with such advection, MPM can keep sharp interface without smearing them. MPM also avoids distortion and entanglement of meshes associated with Lagrange methods in the case of a large acceleration. The present work deals with this numerical instability by transferring the force away from the small mass nodes in a manner consistent with the governing equations for the fluid and solid is employed, and a volume-of-fluid idea is employed to describe a multi-component geometry. The solid stress is defined in Eulerian frame by using a left Cauchy-Green deformation tensor, and the temporal change in the solid deformation is described by updating the tensor. Without numerical diffusion associated with such advection, MPM can keep sharp interface without smearing them. MPM also avoids distortion and entanglement of meshes associated with Lagrange methods in the case of a large acceleration. The present work deals with this numerical instability by transferring the force away from the small mass nodes in a manner consistent with the errors of the original MPM calculation. This treatment significantly improves the stability of multi-phase flow simulation using MPM. The numerical cost of this algorithm is negligible considering the computational time saved by the significantly improved time step size. We provide comparisons between the results calculated with and without this improvement to MPM. Release number: LA-UR 10-05193.

9:57AM GU.00010 Material point method to fluid-structure interactions, DUAN ZHANG, XIA MA, PAUL GIGUERE, Los Alamos National Laboratory, CARTABLANCA TEAM — Fluid-structure interactions are common in nature and in engineering practice. Numerical simulation of these phenomena has been difficult not only because of the need to track interfaces, but also because of the need to model material interactions on the interface region. Furthermore, if the solid material is a porous material, material interactions occur not only on the interface region, but also inside the body of the materials. Recently, this type of problems has been studied using a continuous multiphase flow theory. Material point method (MPM) has been found to be a handy tool for these calculations. The material point method is an advanced version of the particle in cell (PIC) method. Recent developments of the method have significantly improved its stability and accuracy for this type of calculations. The use of the method to fluid-structure interaction problems has produced many promising results. In this talk we will briefly introduce the recent developments of MPM and shown examples of its applications.
8:00AM GV.00001 Renormalized transport of inertial particles. MARCO MARTINS AFONSO, IMFT, INPT - UPS - CNRS, Toulouse (France), ANTONIO CELANI, CNRS - Inst. Pasteur, Paris (France), ANDREA MAZZINO, CNISM - INFN - Univ. Genova (Italy), Piero Olla, ISAC-CNRS - INFN, Cagliari (Italy) — We study how an imposed flow (laminar or turbulent) modifies the transport properties of inertial particles, namely their terminal velocity and effective diffusivity. Such quantities are investigated by means of analytical and numerical computations, as functions of the control parameters of both flow and particle; i.e., density ratio, inertia, Brownian diffusivity, gravity (or other external forces), turbulence intensity, compressibility degree, space dimension, and geometric/temporal properties. The complex interplay between these parameters leads to the following conclusion of interest in the realm of applications: any attempt to model sedimentation processes (or, equivalently, floater transport by surface winds) cannot avoid taking into account the full details of the flow field and of the inertial particles.

8:13AM GV.00002 Particulate Flow over a Backward Facing Step Preceding a Filter Medium, FRANK CHAMBERS, KRISHNA RAVI, Oklahoma State University — Computational Fluid Dynamic predictions were performed for particulate flows over a backward facing step with and without a filter downstream. The carrier phase was air and the monodisperse particles were dust with diameters of 1 to 50 microns. The step expansion ratio was 2.1, and the filter was located at 4.25 and 6.75 step heights downstream. Computations were performed for Reynolds numbers of 6550 and 10000. The carrier phase turbulence was modeled using the k-epsilon RNG model. The particles were modeled using a discrete phase model and particle dispersion was modeled using stochastic tracking. The filter was modeled as a porous medium, and the porous jump boundary condition was used. The particle boundary condition applied at the walls was “reflect” and at the filter was “trap.” The presence of the porous medium showed a profound effect on the recirculation zone length, velocity profiles, and particle trajectories. The velocity profiles were compared to experiments. As particle size increased, the number of particles entering the recirculation zone decreased. The filter at the farther downstream location promoted more particles becoming trapped in the recirculation zone.

8:26AM GV.00003 ABSTRACT WITHDRAWN

8:39AM GV.00004 Numerical simulation of particle laden coaxial turbulent jet flows1, KUMARAN KANNAYAN, REZA SADR, Texas A&M at Qatar — The study of coaxial turbulent particle laden jets has been of interest due to its importance in many applications such as industrial burners, and mixing devices. The addition of the second phase to the continuous phase jet can change the already complicated flow pattern and turbulent characteristics of the jets. Albeit the vast research efforts that have been devoted to understand such phenomena, demand for detailed investigation of particle laden flows remains an active area of research. The advent of laser diagnostics has helped to quantify the myriad details of the jet flow fields in more details. In parallel computational fluid dynamics (CFD) can provide additional information by further investigating such flows with an acceptable level of accuracy. In this work, numerical simulations results are presented for the flow and turbulent characteristics of a coaxial jet with and without the dispersed phase. The results are compared with the experimental data measured using Molecular Tagging Velocimetry diagnostic technique. The key objective of this work is to undermine the flow field details that are difficult if not impossible to measure.

1Supported by QSTP.

8:52AM GV.00005 A-priori reconstruction of ideal stochastic forcing for particle motion in turbulent flow, MARIA VITTORIA SALVETTI, Dept. Aerospace Engineering, University of Pisa, SERGIO CHIBBARO, D’Alembert Institute, Pierre et Marie Curie University, FEDERICO BIANCO, CRISTIAN MARCHIOLI, ALFREDO SOLDATI, Dept. Energy Technologies, University of Udine — One issue associated with the use of Large-Eddy Simulation (LES) to study the dispersion of small inertial particles in turbulent flow is the accuracy with which particle statistics and concentration can be reproduced. The motion of particles in LES fields may differ significantly from that observed in experiments or Direct Numerical Simulation (DNS) because the force acting on particles is not well estimated when only the filtered fluid velocity is available, and because errors accumulate in time leading to progressive trajectories divergence. We identify herein an Ideal Forcing (IF) such that trajectories of individual particles moving in a-priori LES fields in turbulent channel flow coincide with particle trajectories in a DNS. The objective is to compute PDF and statistical moments of IF to possibly identify a stochastic process from which IF could be extracted and then used as closure model for the particle motion equations.

9:05AM GV.00006 Eulerian-Lagrangian Simulations of Three-Dimensional Turbulent Riser Flows, JESSE CAPECELATRO, University of Colorado Boulder, PERRINE PEPIOT, National Renewable Energy Lab, OLIVIER DESJARDINS, University of Colorado Boulder, UNIVERSITY OF COLORADO BOULDER TEAM, NATIONAL RENEWABLE ENERGY LAB COLLABORATION — Particle suspended flows in vertical risers are used in many industries in the form of circulating fluidized beds. Applications include gasification/pyrolysis for biofuel conversion, coal combustion, and fluid catalytic cracking. Experimental studies have shown riser flows to be unsteady with large particle concentration fluctuations. Regions of dense and packed particle dispersion refers to clusters forming, which greatly affect the overall flow behavior and mixing properties. Because the solid phase is opaque, quantitative experimental results are limited, and therefore computational fluid dynamics (CFD) is used here to simulate 3D riser flows. The gas phase is solved with a high-order fully conservative finite difference code called NGA, tailored for turbulent flow computation. Lagrangian tracking is used to solve the particle phase. Statistics are computed for both the gas and particle phases, along with characteristic cluster sizes, shape, and velocities. Inter-particle collisions are considered and shown to affect the clustering behavior of the flow.

9:18AM GV.00007 Numerical prediction of pollutant dispersion into ABL; a Lagrangian approach using LES, MICHAEL MORIKONE, University of Central Florida, STEFAN LLEWELLYN SMITH, University of California, San Diego, MARCEL ILIE, University of Central Florida — Air pollution is one of the major environmental challenges facing humankind today. The accurate prediction of fate and transport of pollutants into the atmospheric boundary layer would improve the health quality and duration of human life, and thus is of critical importance. The pollutants are transported along the wind direction, but it is the atmospheric turbulence that determines the dispersion of the pollutants. An efficient Eulerian-Lagrangian particle dispersion algorithm for the prediction of pollutant dispersion in the atmospheric boundary layer (ABL) is proposed. The volume fraction of the dispersed phase is assumed to be small enough such that particle-particle collisions are negligible and properties of the carrier flow are not modified. With the examination of dilute systems only the effect of turbulence on particle motion has to be taken into account (one-way coupling). With this assumption the continuous phase can be treated separately from the particulate phase. The continuous phase is determined by large-eddy simulation in the Eulerian frame of reference whereas the dispersed phase is simulated in a Lagrangian frame of reference. The results of the present study indicate that the particle shape and size influences the particle dispersion.

9:31AM GV.00008 Coherent vortex interaction with a particle-laden turbulent boundary layer. FERNANDO MORALES, Arizona State University, IFTEHAR NAQAVI, Queen’s University, Kyle SQUIRES, Arizona State University, UGO PIOMELLI, Queen’s University — The focus of the current investigations is numerical modeling of particle entrainment in turbulent boundary layers with and without coherent vortices superimposed on the background flow. Simulations are performed using an Euler-Lagrange method in which a fractional-step approach is used for the fluid and solid phases, where the particulate phase advanced using Discrete Particle Simulation. The first flow field models a rotor wake comprised of gradually introduced coherent vortices into a turbulent boundary layer. The second flow is a turbulent boundary layer without vortices to discriminate and characterize the effect of the vortex structures on the dispersed phase properties. The third case models interaction of a coherent vortex introduced into a stagnant bed of particles. The simulations are performed with two groups of particles having different densities both of which display strong vortex-particle interaction close to the source location, and with mixing of the particles into the boundary layer downstream. Visualizations and statistical descriptors quantify the strong effect of the coherent vortex structures on dispersed phase properties.
9:44AM GV.00009 Large-eddy simulation of turbulent collision of heavy particles in isotropic turbulence, GUODONG JIN, GIOWEI HE, State Key Laboratory for Nonlinear Mechanics, Institute of Mechanics, Chinese Academy of Sciences — The small-scale motions relevant to the collision of heavy particles represent a challenge to LES of turbulent particle-laden flows. We examine the capability of the LES method to predict the collision-related statistics such as the collision rate for a wide range of particle Stokes numbers. It is shown that, without the SGS motions, LES cannot accurately predict the particle-pair statistics for heavy particles with small and intermediate Stokes numbers. The errors from the filtering operation and the SGS model are evaluated separately using the filtered-DNS (FDNS) and LES flow fields. The errors increase with the filter width and have nonmonotonic variations with the particle Stokes numbers. It is concluded that the error due to filtering dominates the overall error in LES for most particle Stokes numbers. It is found that a particle SGS model must include the effects of SGS motions on the turbulent collision of heavy particles for $St < 3$. For more details please refer to Phys. Fluids 22, 055106 (2010).

9:57AM GV.00010 Effects of Inter-Particle Collisions and Two-Way Coupling on Particle Deposition Velocity in a Turbulent Channel Flow, HOJJAT NASR, Carnegie Mellon University, GOODARZ AHMADI, JOHN MCLAUGHLIN, Clarkson University — This study was concerned with the effect of particle-particle collisions and two-way coupling on the particle deposition velocity in a turbulent channel flow. The time history of the instantaneous turbulent velocity vector was generated by the two-way coupled direct numerical simulation (DNS) of the Navier-Stokes equation via a pseudospectral method. The particle equation of motion included the Stokes drag, the Saffman lift, and the gravitational forces. The effect of particle-particle collisions on the fluid was included in the analysis via a feedback force that acted on the fluid on the computational grid points. Several simulations for different particle relaxation times and particle mass loading were performed, and the effects of the inter-particle collisions and two-way coupling on the particle deposition velocity, fluid and particle fluctuating velocities, particle normal mean velocity, and particle concentration were determined. It was found that when particle-particle collisions were included in the computation, the particle deposition velocity increased. When the particle collision was neglected but the particle-fluid two-way coupling was accounted for, the particle deposition velocity decreased slightly. For the four-coupling case, when both inter-particle collisions and two-way coupling effects were taken into account, the particle deposition velocity increased. Comparisons of the present simulation results with the available experimental data and earlier numerical results are also presented.

Monday, November 22, 2010 8:00AM - 10:10AM – Session GW Surface Tension I

8:00AM GW.00001 Capillary wrinkling of elastic membranes, DOMINIC VELLA, University of Cambridge, MOKHTAR ADDA-BEDIA, LPS de l’ENS, Paris, ENRIQUE CERDA, Universidad de Santiago, Chile — We present a physically-based model for the deformation of a floating elastic membrane caused by the presence of a liquid drop. Starting from the equations of membrane theory modified to account for the three surface tensions in the problem, we show that the presence of a liquid drop causes an azimuthal compression over a finite region. This explains the origin of the wrinkling of such membranes observed recently [J. Huang et al., Science 317, 650 2007] and suggests a single parameter that determines the extent of the wrinkled region. While experimental data supports the importance of this single parameter, our theory under-predicts the extent of the wrinkled region observed experimentally. We suggest that this discrepancy is likely to be due to the wrinkling observed here being far from threshold and discuss other, related, geometries.

8:13AM GW.00002 A Computational Study of Surfactant Effects in the Bretherton Problem, METIN MURADOGLU,UFUK OLGAC, GOKALP GURSEL, Koc University — The finite-difference/front-tracking method developed by Muradoglu and Tryggvason (2008) is used to study the motion and deformation of a large bubble moving through a capillary tube in the presence of both insoluble and soluble surfactants. Emphasis is placed on the effects of surfactant on the liquid film thickness between the bubble and tube wall. The numerical method is designed to solve the evolution equations of the interfacial and bulk surfactant concentrations coupled with the incompressible Navier-Stokes equations. A non-linear equation of state is used to relate interfacial surface tension to surfactant concentration at the interface. The method is validated for simple test cases and the computational results are found to be in a good agreement with the analytical solutions. The film thickness is first computed for the clean bubble case and the results are compared with the lubrication theory in the limit of small capillary numbers, i.e., $Ca \ll 1$. Finally the method is used to simulate the effects of insoluble and soluble surfactants on the film thickness for a wide range of governing non-dimensional numbers.

8:26AM GW.00003 Vapor stabilizing surfaces for superhydrophobicity, NEELESH PATANKAR, Northwestern University — The success of rough substrates designed for superhydrophobicity relies crucially on the presence of air pockets in the roughness grooves. This air is supplied by the surrounding environment. However, if the rough substrates are used in enclosed configurations, such as in fluidic networks, the air pockets may not be sustained in the roughness grooves. In this work a design approach based on sustaining a vapor phase of the liquid in the roughness grooves, instead of relying on the presence of air, is explored. The resulting surfaces, referred to as vapor stabilizing substrates, are deemed to be robust against wetting transition even if no air is present. Applications of this approach include low drag surfaces, nucleate boiling, and dropwise condensation heat transfer, among others.

8:39AM GW.00004 Dynamic surface tension effects from molecular dynamics simulations, ALEX LUKYANOV, ALEXEI LIKHTMAN, University of Reading — Effects of dynamic surface tension have been studied in a model system using molecular dynamics simulations. The model system has been made of an artificially expanding liquid droplet, with the rate of change of the external surface area being comparable with the gas-liquid interface formation characteristic time, obtained from the estimates of macroscopic theories. The size of the liquid droplet has been chosen to have about 5,000-7,000 identical chain molecules, each having between 10-20 beads, to obtain well developed and separated the bulk and the surface phases. The methodology of surface tension evaluation has been verified against the Laplace Law in a stationary state of the liquid drop. The results of the molecular dynamics simulations will be discussed in comparison with the estimations obtained from macroscopic experimental data. We used a sharp interface formation theory for different chain length of molecules and strength of intermolecular interactions.

8:52AM GW.00005 The Dynamics of Droplets and Holes in Thin Surfactant Films, KAREN DANIELS, NC State University, KALI ALLISON, Harvey Mudd College, JONATHAN CLARIDGE, University of Washington, RACHEL LEVY, Harvey Mudd College, ELLEN PETERSON, MICHAEL SHEARER, NC State University, WYNN VONNEGUT, JEFFREY WONG, Harvey Mudd College — We perform quantitative measurements of the spreading of an insoluble surfactant on a thin layer of glycerin, starting from either a droplet or hole (anti-droplet) configuration. We make direct measurements of both the radial height profile and the spatial distribution of the fluorescently-tagged surfactant during the spreading process. Remarkably, the surfactant dynamics are quite different for the two cases. We compare these experimental results to numerical solutions using lubrication theory for thin films.
Recently we have demonstrated pinned-contact, coupled droplet pairs of aqueous ferrofluids in air that can form electromagnetically-activated capillary switches and oscillators. The great variety of available ferrofluids, however, enables the use of immiscible oil-based ferrofluid droplets in a water environment to obtain the same behavior. Such immersed ferrofluid oscillators exhibit natural frequencies (for 5 mm devices) of about 10 Hz. Here we report on the observation of a gradual increase in the resonant frequency of the system in time. Experimental observations suggest that the drift in the natural frequency is a consequence of changes occurring at the ferrofluid/water interface. The interfacial structure of such a complex system (water, oil, surfactant, iron particles) is examined along with its evolution in time, using various microscopy techniques.

8:18AM GW.00007 Forces to Dislodge Rotating Sessile and Pendant Annular Rivulets, P.D. WEIDMAN, University of Colorado, Boulder, C.P. MALHOTRA, TCS Innovation Labs, Pune, India — In a recent PRL Tadmor, et al (2009) measured the lateral adhesion force on sessile and pendant drops (of equal volume placed at R = 100 cm) on a rotating flat surface (treated mica substrate) and found that more force is required to radially displace the pendant drops. This was explained as enhanced chemical interaction between liquid and solid molecules when the drop is pendant compared to sessile. We take the view it is primarily static advancing and receding contact angles that govern the movement of the drops. This is shown by a simple model where the isolated drop is replaced by a thin axisymmetric rivulet. For realistic advancing and receding contact angles of water on anodized aluminum, computations performed show the existence of four distinct regions governing drop movement. The three regions found at small radii give way at R = 2.0 cm to the final fourth region where drop movement depends on the advancing contact angle for both sessile and pendant drops, here the pendant drops require a larger radial force to dislodge, in agreement with the measurements of Tadmor, et al. Simulations more closely mimicking the experiments of Tadmor, et al will be presented.

9:31AM GW.00008 Filtration by eyelashes, KRISHNA VISTARAKULA, MIKE BERGIN, DAVID HU, Georgia Institute of Technology — Nearly every mammalian and avian eye is rimmed with lashes. We investigate experimentally the ability of lashes to reduce airborne particle deposition in the eye. We hypothesize that there is an optimum eyelash length that maximizes both filtration ability and extent of peripheral vision. This hypothesis is tested using a dual approach. Using preserved heads from 36 species of animals at the American Museum of Natural History, we determine the relationship between eye size and eyelash geometry (length and spacing). We test the filtration efficacy of these geometries by deploying outdoor manikins and measuring particle deposition rate as a function of eyelash length.

9:44AM GW.00009 The hydrodynamics of ink writing, JUNG CHUL KIM, HO-YOUNG KIM, Seoul National University — When one writes on the paper with a pen, the ink spreads on the porous hydrophilic solid surface leaving a trail whose width depends on the pen speed and the physicochemical properties of the ink and of the paper. Here we mathematically describe the spreading profile of the ink, which is modeled to be a liquid from a tube wicking through a rough hydrophilic surface (micropillar arrays). By balancing the capillary forces that drive the liquid flow with the viscous forces exerted by the forest of micropillars, we obtain the rate of ink spreading. Considering the liquid spreading and the pen translation that occur simultaneously, we predict the frontal shape and the final width of a line that is drawn by the pen. We performed experiments using various kinds of liquids and different dimensions of micropillar arrays that are coated with Si-DLC film and treated with oxygen plasma. The theory and experiment are shown to be in excellent agreement.

9:57AM GW.00010 Marangoni patterns, N. NIRMAL THYAGU, Postdoctoral associate, Dept. of Biomedical Engineering, Rutgers University, Piscataway, N.J., EVELYN STROMBÖM, DANIÈL PALUMBO, undergraduate, CARLOS CAICEDO, TROY SHINBROT, Professor of Biomedical Engineering, Rutgers University, Piscataway, N.J. — We study Marangoni patterns that emerge when common food dye is dropped into a dish of shallow water. These patterns consist of tendrils and spots that sharpen over time before eventually fading. We demonstrate that the patterns can be modeled using coupled reaction-diffusion equations, where the “reaction” terms appear due to a nonlinear dependence of surface tension on dye concentration. We show using a spatio-temporal metric that these patterns are distinct from previously described Turing patterns.

Session GX Compressible Flows I
Hyatt Regency Long Beach Regency D

8:00AM GX.00001 ABSTRACT WITHDRAWN

8:13AM GX.00002 Observable Divergence and Observable Euler Equations for Shocks and Turbulence, KAMRAN MOHSENI, University of Colorado at Boulder — Both turbulence and shock formation in inviscid flows are prone to high wave number mode generations. This continuous generation of high wavenumbers results in an energy cascade to ever smaller scales in turbulence and/or creation of shocks in compressible flows. This high wavenumber problem is often remedied by the addition of a viscous term in both compressible and incompressible flows. A regularization technique for the Burgers equation (Norgard and Mohseni 2008) was recently reported. This inviscid regularization was extended to one-dimensional compressible Euler equations in 2010 (Norgard and Mohseni 2010). This investigation presents a formal derivation of these equations from basic principles. Our previous results are extended to multidimensional compressible Euler equations. We define a new observable divergence based on fluxes calculated from observable quantities at a desired scale. An observable divergence theorem is then proved and applied in the derivation of the regularized equations. It is shown that the derived equations reduce to incompressible Leray flow model in the limit of incompressibility. It is expected that this technique simultaneously regularizes shocks and turbulence for compressible and incompressible Euler equations. Finally, numerical simulations are presented for the compressible one-dimensional observable Euler equations.

Supported by the AFOSR.

8:26AM GX.00003 A Numerical Study of the unstart event in a supersonic engine, IK JANG, RENE PECKIN, PARVIZ MOIN, Stanford University — The objective of this study is to assess the capability of an unsteady Reynolds averaged Navier-Stokes (URANS) method to predict the unstart phenomenon in a scramjet engine. Both started and unstarted Mach 5 flows in the inlet/isolator part of a scramjet engine were numerically investigated. The unstart event is initiated by raising a mechanical flap located at the downstream of the isolator, and the motion of the flap was simulated by an immersed boundary method. The simulation results were compared with a series of experiments (Wagner et al., AIAA paper, 2007, 2008, 2009) and a hybrid LES/RANS study (Boles et al., AIAA paper, 2009) performed on the same geometry and flow conditions. The critical angle of the flap to initiate the unstart event as well as the pressure distribution on the wall of the isolator in the started flow are in good agreement with the experiment and the simulation of Boles et al. On the other hand, the upstream moving shock speed is not well predicted and the cause of discrepancy is the subject of the ongoing study.

Supported by the PSAAP program of DOE.
8:39AM GX.00004 Compressibility effects in planar wakes , JEAN-PIERRE HICKEY, Royal Military College of Canada,
FAZLE HUSSAIN, University of Houston, XIAOHUA WU, Royal Military College of Canada — Far-field, temporally evolving planar wakes are studied by DNS to evaluate the effect of compressibility on the flow. A high-order predictor-corrector code was developed and fully validated against canonical compressible test cases. In this study, wake simulations are performed at constant Reynolds number for three different Mach numbers: $Ma = 0.2, 0.8$ and 1.2. The domain is doubly periodic with a non-reflecting boundary in the cross-flow and is initialized by a randomly perturbed laminar profile. The compressibility of the flow modifies the observed structures which show greater three-dimensionality. A self-similar period develops in which the square of the wake half-width increase linearly with time and the Reynolds stress statistics at various times collapse using proper scaling parameters. The growth-rate increases with increasing compressibility of the flow: an observation which is substantiated by experimental results but is in stark contrast with the high-speed mixing-layer. As the growth-rate is related to the mixing ability of the flow, the impact of compressibility is of fundamental importance. Therefore, we seek an explanation of the modified growth-rate by investigating the turbulent kinetic energy equation. From the analysis, it can be conjectured that the pressure-strain term might play a role in the modified growth-rate.

8:52AM GX.00005 Energy-Pressure-Velocity Filtered Mass Density Function , MEHDI B. NIK, PEYMAN GIVI, University of Pittsburgh, CYRUS MADNIA, University at Buffalo (SUNY), STEPHEN B. POPE, Cornell University — A new methodology termed “energy-pressure-velocity filtered mass density function” (EPV-FMDF) is developed for large eddy simulation of high speed turbulent flows. This is an extension of the previously developed “velocity filtered density function” (V-FDF) method [1] in low speed flows. To account for the effect of compressibility, the formulation includes two additional thermodynamic variables: the pressure and the internal energy. This is the most general form of the FDF for high speed flow simulations. The EPV-FMDF is obtained by solving its transport equation, in which the effects of convection appear in a closed form. The unclosed terms are modeled in a fashion similar to that in RANS-PDF methods. The modeled EPV-FMDF transport equation is solved by a Lagrangian Monte Carlo method and is employed for LES of a temporally developing mixing layer at several values of the convective Mach number. The predicted results are assessed by comparison with direct numerical simulation (DNS) data.


9:05AM GX.00006 A PIV Study of Compressibility in a Turbulent Subsonic Axisymmetric Jet , ISAAC CHOUTAPALLI, University of Texas - Pan American — An experimental study has been undertaken to study the effect of nozzle exit Mach number on the flow field characteristics of a turbulent round free jet. Particle Image Velocimetry (PIV) is used as a means of measurement. The Mach numbers considered are 0.30, 0.65 and 0.80. The measurements show that the centerline mean velocity variation, the radial profiles of mean velocity and turbulence quantities within the shear layer can be collapsed onto single curves over the range of Mach numbers considered. The data further shows that it is possible to represent the shear layer mean velocity profile for the turbulent free jet in the subsonic Mach number range with a single analytical expression.

9:18AM GX.00007 Shock waves in dense hard disk media: molecular dynamic and continuum descriptions , NICK SIRMAS, MARIAN TUDORACHE, MATEI I. RADULESCU, University of Ottawa — Mediums composed of a system of colliding hard disks (2D) or hard spheres (3D) serve as good approximations to the molecular structure of gases, liquids and granular media. In the present study, the propagation of piston driven shock waves in a two-dimensional hard-disk medium is studied at both the continuum and discrete particle level descriptions. For the continuum description, closed form analytical expressions for the shock Hugoniot and shock jump conditions were obtained using the approximate Helfand equation of state. The predictions were found in excellent agreement with calculations using the Event Driven Molecular Dynamic method involving 30,000 particles over the entire range of compressibility spanning the dilute ideal gas, liquid and solid phases. In all cases, the energy imparted by the piston motion to the thermalized medium behind the propagating shock is found to be quasi-independent of the medium packing fraction, with a correction vanishing with increasing shock Mach numbers.

9:31AM GX.00008 Shock wave dynamics in a water-filled log-spiral duct , CHUANXI WANG, VERONICA ELIASIASSON, University of Southern California — Both numerical simulations and experiments of converging shock waves in a water-filled log-spiral duct have been performed. A log-spiral shape is considered as one of the “worst-case scenarios,” because this particular shape minimizes reflections from the surrounding walls. The simulations were performed using the Overture suite, a package for solving partial differential equations on curvilinear overlapping grids using adaptive mesh refinement. A coupled fluid-solid solver was used to simulate both the water and the surrounding solid domains. Results show that the fluid-structure interaction indeed changes the shock dynamics as compared to using fluid domain with rigid boundary conditions. In the experiments, a projectile from a gas gun impacts the water-filled sample and generates a shock wave, which is then captured by a high-speed schlieren system. Comparison between the experiments and the simulations will be presented, and the results can benefit the design of marine structures with converging sections subjected to dynamic loading events.

9:44AM GX.00009 Shock Waves in Dense Fluids: An Anisotropic Temperature Theory with Delays , FRANCISCO J. URIBE, Universidad Autonoma Metropolitana, WILLIAM G. HOOVER, CAROL G. HOOVER, Ruby Valley Research Institute — In this work we analyze a recent phenomenological hydrodynamic theory proposed recently by Hoover et al. [arXiv:1005.1525v1] to study shock waves in dense fluids. The theory incorporates anisotropic temperature and relaxation for the fluxes and temperature, following the ideas by Maxwell, Cattaneo, and Krook. For the steady case we analyze the points at which the vector field is infinite (singularities of the field) and obtain conditions for non-existence of a shock profile connecting the two relevant equilibrium points (non-existence of heteroclinic connections). The conditions for non-existence of heteroclinic orbits are then related to the nature of the critical (equilibrium) points and the bifurcations of the system. The results are tested using several numerical methods to solve the hydrodynamic equations for the non-steady and steady problems.

9:57AM GX.00010 The Structure of Weak Shock Waves in Water , ROY BATY, Los Alamos National Laboratory,
DON TUCKER, University of Utah, CARL HAGELBERG, Los Alamos National Laboratory — This talk presents solutions of the Navier-Stokes equations that model weak shock waves in water. One-dimensional jump functions are computed to describe the viscous microstructure of hydrodynamic shocks, which are approximately isentropic. The Tate and Grueneisen equations of state are applied separately with the conservation laws to derive the flow microstructure for shock compressions and pressures up to 1.3 and 20.0 kbars, respectively. The Navier-Stokes equations are integrated along characteristic lines to compute the shock wave thickness. On characteristic lines, the shock wave jump functions reduce to integral equations. The Tate and Grueneisen equations of state yield similar, strictly monotonically increasing, shock wave microstructures. Moreover, the non-dimensional shock wave thicknesses predicted by these equations of state as a function of compression are very similar.

Monday, November 22, 2010 8:00AM - 10:10AM – Session GY Instability: Interfacial and Thin Film IV Hyatt Regency Long Beach Regency E
8:00AM GY.00001 An Energy Approach to Capillary-Driven Thin Film Folding , HUAN LI, K. JIMMY HSIA, SASCHA HILGENFELDT, University of Illinois at Urbana-Champaign — Capillary-driven self-assembly methods provide a promising tool to fabricate three-dimensional, micro- or millimeter scale structures. Recently, we explored the self-assembly of 3D photovoltaic devices from Si thin films through equilibrium considerations of fluid-solid interactions (Guo, et al. 2009, Li, et al. 2010). In the present study, an alternative approach, the minimization of the total free energy is employed to investigate the interactions between fluid droplet and a flexible thin film. Variation of a 2D energy functional, comprising the surface energy of the fluid and the bending energy of the thin film, yields governing equations and boundary conditions. Through direct simulations with Surface Evolver (Brake 2008), the shape of the droplet and the thin film at the equilibrium state is obtained. A critical thin film length necessary for complete enclosure of the fluid droplet, and thus successful device self-assembly, is determined and compared with the experimental study of Guo et al. (2009). Augmenting the formalism, we obtain an upper bound of the thin film length, beyond which gravity is dominant. The current 2D study can be extended into 3D. Critical parameters obtained from these analyses can be used to guide device fabrication and manufacturing.

8:13AM GY.00002 Thin films: instabilities, waves, and dewetting , TE-SHENG LIN, LOU KONDIC, New Jersey Institute of Technology — We study free surface instabilities of spreading thin films exposed to destabilizing body force (gravity) on partially wetting substrates. For completely wetting films on inverted substrates, we have uncovered rich structure of convective and absolute instabilities which evolve due to contact line presence. In this talk, we will concentrate on partially wetting case, where additional destabilizing component of disjoining pressure may lead to significant modifications of the instabilities discovered for the complete wetting case. In particular, we consider the interplay between different destabilizing mechanisms to discuss free surface instabilities which lead to dewetting, in contrast to those which do not. We conclude by presenting preliminary results of three dimensional simulations showing the interplay between free surface and fingering type of instabilities.


8:26AM GY.00003 Experimental Analysis of the Instability and Breakup of an Annular Liquid Sheet with Axial Co-flow , DANIEL DUKE, JULIO SORIA, DAMON HONNERY, Laboratory for Turbulence Research in Aerospace & Combustion, Monash University — A novel application of a correlation-based velocimetry technique combined with high speed imaging has been employed to measure the interfacial stability properties of a thin annular liquid sheet with dual co-flowing axial gas streams undergoing aerodynamic breakup. The measurement of instabilities in thin liquid sheets has traditionally been limited due to the challenging complexities of taking measurements in the near-nozzle region where instability amplitudes are very small. Agreement with theoretical stability analysis has also proven elusive. The application of new techniques permits quantitative velocity measurement of with significantly improved resolution, for a more detailed analysis of the instability properties. Latest results are shown for the effects of inner and outer co-flow Re, sheet thickness and liquid Re on the aerodynamically driven instabbily. Improved agreement with theoretical stability analysis is also shown. Improved techniques may afford a better understanding of the complex dynamics of sheet atomisation.

8:39AM GY.00004 Wavy regimes of a film flow down a slip inclined plane1, ARGHYA SAMANTA, CHRISTIAN RUYER-QUIL, Université Pierre et Marie Curie , BENOIT GOYEAU, Ecole Centrale de Paris — Consider a 2D viscous incompressible liquid film flow down a slip inclined plane under the action of gravitational force. Two coupled depth averaged equations are derived in terms of the flow rate $q(x,t)$ and the film thickness $h(x,t)$ with the framework of boundary layer approximations together with weighted-residual technique. Linear stability analysis of the averaged equations show good agreement with the results of Orr-Sommerfeld analysis of linearized basic equations. At small values of Reynolds number, presence of slip length is destabilizing whereas it becomes stabilizing at moderate values of Reynolds number. This phenomena indicates the nontrivial stabilizing effect of slip length on the primary instability. In the nonlinear regime, the influence of slip length has been investigated through traveling-wave solutions of the averaged equations. Comparisons with direct numerical simulations of Navier-Stoke’s equations also show good agreement.

1Supported by Marie Curie ITN Multiflow network No 214919.

8:52AM GY.00005 Wavy regime of a viscoplastic film flow1 , SYMPHONY CHAKRABORTY, CHRISTIAN RUYER-QUIL, Université Pierre et Marie Curie , FAST, Campus Universitaire, 91405 Orsay, France, BHAIBAN S. DANDAPAT, Sikkim Manipal Institute of Technology, Majitar, Rango, 737 132, East Sikkim, India — We consider a power-law fluid flowing down an inclined plane under the action of gravity. The divergence of the viscosity of a shear-thinning fluid at zero strain rate is taken care of by introducing a Newtonian plateau at small strain rate. Applying a weighted residual approach, a two-equations model is formulated in terms of two coupled evolution equations for the film thickness $h$ and the local flow rate $q$ within the framework of lubrication theory. The model accounts for the streamwise diffusion of momentum. Consistency of the model is achieved up to first order in the film parameter for inertia terms and up to second order for viscous terms. Comparison to Orr-Sommerfeld stability analysis and to DNS show convincing agreement in both linear and nonlinear regimes. In the case of shear-thinning fluids, lowering the power index has a non-trivial effect on the primary instability of the film: the threshold of the instability occurs at a smaller Reynolds number but the range of instable wavenumber is also reduced. In the nonlinear regime, we have evidenced a subcritical bifurcation of the traveling-wave solutions from marginal stability conditions.

1Supported by Marie Curie ITN Multiflow network No 214919.

9:05AM GY.00006 Rupture of thin liquid films with Plateau borders1 , LUCIEN BRUSH, ALAN MCINTYRE, University of Washington, STEVEN ROPER, University of Glasgow — In metal foams there is fluid flow from a lamella into the Plateau borders resulting in lamellar thinning. Since surfactants are not used to slow the flow, instability of a lamella quickly leads to rupture, bubble coalescence and overall coarsening of the foam. This talk presents the results of numerical calculations of the rupture process of a lamellar film with Plateau borders in a gas-liquid metallic foam. The numerical calculations show the evolution of a lamella from the initiation of an instability up to the time just prior to rupture. Rupture times and locations are monitored as a function of the Plateau border radius of curvature. The effect of symmetry-breaking configurations in which a lamella spans two Plateau borders having different radii of curvature shows that the location of rupture can be near the thin film - Plateau border junction. Solutions at late times are compared to the similarity solutions for the case of free films without Plateau borders.

1Support from NSF Grant No. CMMI-0827101 is acknowledged.

9:18AM GY.00007 An ADI Scheme for Particle-Laden Thin Film Flow in 2D1 , MATTHEW MATA, UCLA — We derive a semi-implicit numerical scheme for a lubrication model of particle-laden thin film flow in two dimensions. The scheme relies on an ADI process to handle the higher-order and advection terms, and an iterative procedure is utilized within each time step in order to improve the quality of the solution and the size of the time step. We compare computational results to laboratory experiments involving mixtures of silicon oil and glass beads on an incline. The results of the simulation agree qualitatively with the experiment and suggest some possible improvements to the model to achieve a quantitative match.

1The research is supported by NSF grant DMS-1048840 and a UC Lab Fees Research Grant.
9:31AM GY.00008 Instability and rupture of thin liquid films on structured surfaces, ELIZAVETA GATAPOVA, Institute of Thermophysics, SB RAS (Russia), VLADIMIR AJAEV, Southern Methodist University, OLEG KABOV, Institute of Thermophysics, SB RAS (Russia) — We investigate stability and break-up of a thin liquid film on a solid surface under the action of disjoining pressure. The solid surface is structured by parallel grooves. Air is trapped in the grooves under the liquid film. Our mathematical model takes into account the effect of slip due to presence of menisci separating the liquid film from the air inside the grooves, the deformation of these menisci due to local variations of pressure in the liquid film, and non-uniformities of the Hamaker constant. Rupture time is found to decrease due to the presence of the grooves. It is shown that simplified descriptions of the film dynamics, e.g. using the standard formulas for effective slip, can lead to significant deviations from the behaviour seen in our simulations. A new regime of self-similar behaviour is found, which is different from the known solutions for films on flat solid substrates and free liquid films.

9:44AM GY.00009 Rupture Limit of Thin Moving Films, JUAN C. PADRINO, DANIEL D. JOSEPH, HYUNJU KIM, University of Minnesota — The rupture of a thin film in another fluid is studied including the effects of disjoining pressure. The study considers the linear stability of a moving viscous film in a motionless inviscid fluid and of a stagnant viscous film in a motionless viscous fluid. These are analyzed by means of the Navier–Stokes equations and the dissipation approximation based on potential flow. Results reveal that the dissipation method provides a good approximation for the case of a moving film, whereas its predictions are off the mark for the stagnant film case. The thickness of the gap at the trough of Kelvin-Helmholtz waves locates the formation of holes. The wavelength at final collapse is determined by the length of waves at the trough of the corrugated film. The disjoining pressure effects cause very fast break-up for very thin films. These effects influence the cutoff wavenumber. In the limit of small gaps on this corrugated film, the Reynolds and Weber numbers tend to zero with the gap size, the Ohnesorge number increases like the reciprocal of the square root and the Hamaker number like the reciprocal of the square of the gap. The motion of the film does not enter at the point of formation of holes. Moreover, for the most unstable wave, the ratio of the wavelength to film thickness is found to decrease with increasing film thickness.

9:57AM GY.00010 Interplay between phase separation and surface deformations in thin films of binary mixtures, SANTIAGO MADRUGA, Universidad Politecnica de Madrid, FATHI BRIBESH, UWE THIELE, Loughborough University — Films of polymer blends are used in technological applications such as coatings or structured functional layers. The evolution of these films is involved by the coupling of phase separation within the film and surface deformations. We developed a model for films of binary mixtures [1], such as polymer blends, with free evolving surfaces. The model is based on model-H describing the coupled transport of concentration and momentum fields supplemented by boundary conditions at the substrate and free surface. We use the model to analyze the stability of vertically stratified base states of free surface films and the influence of composition in the shape of the films. For purely diffusive transport, an increase in film thickness either exponentially decreases the lateral instability or entirely stabilizes the film. The inclusion of convective transport leads to further destabilization as compared to the purely diffusive case [2]. In addition, we discuss the role of composition for off-critical mixtures on surface deflections.


Monday, November 22, 2010 8:00AM - 10:10AM –
Session GZ Instability: General I Hyatt Regency Long Beach Regency F

8:00AM GZ.00001 Onset of sustained turbulence in pipe flow, KERSTIN AVILA, ALBERTO DE LOZAR, BJÖRN HOF, Max Planck Institute for Dynamics and Selforganization — The onset of turbulence in pipe flow was first investigated by Reynolds more than 125 years ago. The laminar Poiseuille profile is linearly stable, so that the precise Reynolds number at which the flow becomes turbulent depends on the care taken to minimize disturbances in the experimental setup. In order to avoid this setup effects, an external localized perturbation is induced in the laminar flow and the development of the ensuing localized turbulent patch (puff) while traveling downstream is studied. Although it was recently found that the puffs are transient for all Re, puffs may grow and split, leading to a spread of turbulence. Here we analyze the splitting of puffs quantitatively and show that it is the competition between decaying and splitting that determines the onset of sustained turbulence. In an in depth experimental investigation, of more than 100,000 measurements in pipes of up to 3.400 diameters in length, we determine the critical Reynolds number for the onset of sustained turbulence in pipe flow.

8:13AM GZ.00002 Towards uniformly turbulent pipe flow?, MARC AVILA, BJÖRN HOF, Max Planck Institute for Dynamics and Selforganization — Turbulence occurs first in pipes in the form of localized spots of fluctuating but well defined size. These spots may relaminarize, merge or split, giving rise to the large-scale laminar-turbulent patterns of the transitional regime. We report on direct numerical simulations and experiments of the transition between these intermittent flows and uniform turbulence. Here, long periodic pipes of up to 500 diameters are used in order to capture the patterns selected by the flow. A large number of simulations is carried out to statistically demonstrate that the limit of uniform turbulence is only reached asymptotically. In particular, it is shown that the relaminarization probabilities of pipe sections of arbitrary lengths are nonzero.

8:26AM GZ.00003 Edge formation in low-dimensional models of shear transition, NORMAN LEBOVITZ, University of Chicago — Low dimensional models are used to illustrate the nature of an edge state. In these models the edge is the stable manifold of a lower-branched equilibrium point. It come into existence in connection with the birth of a periodic orbit via a homoclinic bifurcation as a parameter (the Reynolds number) increases beyond a critical value. Even for values of the Reynolds number less than this critical value, the structure of the basin boundary is such that edge-like behavior occurs in parts of phase space. It is possible to manufacture dynamical systems for which the edge state disappears for sufficiently large parameter values.

8:39AM GZ.00004 Spatially localized solutions of plane Couette flow1, JOHN GIBSON, University of New Hampshire, TOBIAS SCHNEIDER, Harvard University, JOHN BURKE, Boston University — We examine spatially localized solutions of plane Couette flow: traveling waves and equilibria with finite spanwise extent and periodic streamwise structure. We show that these solutions exist over a wide range of Reynolds numbers, from Re=170 to at least Re=4000, and demonstrate a relationship between the streamwise periodicity of a solution and the range of Reynolds number over which it appears. Some solutions display a diagonal or winding symmetry, suggestively similar to the diagonal bands of structure observed in large-scale simulations by Tuckermann and Barkley.

1Partially supported by NSF grant DMS-0807574.
8:52AM GZ.00005 A Search for Exact Coherent Structures in Transitional Taylor-Couette Flow
  1  DANIEL BORRERO-ECHEVERRY, Center for Nonlinear Science, Georgia Institute of Technology, DONALD R. WEBSTER, School of Civil & Environmental Engineering, Georgia Institute of Technology, RANDALL TAGG, Department of Physics, University of Colorado at Denver, MICHAEL F. SCHATZ, Center for Nonlinear Science, Georgia Institute of Technology — Theoretical and numerical studies have suggested that unstable, exact solutions of the Navier-Stokes equations known as Exact Coherent Structures (ECS) may provide a foundation for a simplified dynamical description of turbulence. We use tomographic particle image velocimetry to measure the velocity field of transitional Taylor-Couette flow (TCF). Specifically, we present spatially and temporally resolved measurements of three-component, three-dimensional velocity fields of turbulent patches that show up when TCF undergoes a subcritical transition to turbulence. This transition occurs when only the outer cylinder rotates and is different from the famous transition driven by centrifugal instabilities. TCF offers the best opportunity to make the connection with current ECS theory since it maintains some of its assumptions (streamwise periodic boundary conditions and plane Couette flow (in the small-gap limit)), but also includes realistic effects (no-slip spanwise boundary conditions).

9:05AM GZ.00006 On the geometry of coexisting edge states for plane Couette flow
  1  LINA KIM, JEFF MOEHLIS, University of California, Santa Barbara — For certain shear flows, it has recently been suggested that the codimension-1 manifolds of an exact coherent structure, called an edge state, can define the boundary which separates trajectories that directly decay to the laminar state from those that become turbulent. This boundary is referred to as the edge of chaos. For a range of aspect ratios for plane Couette flow, distinct edge states can be found using an iterative method. We explore the geometry associated with these coexisting edge states, and the relationship of the edge to the turbulent and laminar dynamics.

9:18AM GZ.00007 Experimental observation of the edge state in pipe flow
  1  ALBERTO DE LOZAR, Max Planck Institute for Dynamics and Self-Organization, FERNANDO MELLIBOVSKY, Universitat Politecnica de Catalunya, BJOERN HOF, Max Planck Institute for Dynamics and Self-Organization — Transition to turbulence in pipe flow is subcritical and therefore laminar and turbulent flows are observed at the same Reynolds number. Recent numerical studies have identified the hyper-surface in phase space which divides trajectories leading to laminar or turbulent state. Surprisingly, a single chaotic attractor (called edge state) controls the flow dynamics on this hyper-surface. It has been suggested that edge state may play an important role for transition to turbulence but up to now there is no experimental evidence to support this claim. Our goal is to look for possible signatures of edge dynamics in decaying turbulence. In a recent paper we demonstrated that turbulence at low Reynolds can be forced to decay. In our experiments we study the flow in this decaying section using two stereo PIV systems enabling us to measure velocities in two planes separated by 6 diameters. We correlate the experimentally measured velocity fields with the numerically calculated edge state. Surprisingly the experiment closely resemble the edge state for 17% of the time. Additionally, the phase velocity in the experiment closely matches the traveling wave solution underlying the edge state.

9:31AM GZ.00008 Eckhaus instability and homoclinic snaking in plane Couette flow
  1  JOHN BURKE, Boston University, JOHN GIBSON, University of New Hampshire, TOBIAS SCHNEIDER, Harvard University — Homoclinic snaking in wide plane Couette channels gives rise to exact solutions of the Navier-Stokes equation which are spatially localized. In this talk, we examine the upper limit of the snaking branches, where the localized states resemble holes of laminar flow embedded in an otherwise regular spatially periodic state. The termination of the snaking branches is related to the Eckhaus instability of the spatially periodic equilibria, but also depends sensitively on the width of the domain.

9:44AM GZ.00009 ABSTRACT WITHDRAWN

9:57AM GZ.00010 A homoclinic tangle at the edge of shear turbulence
  1  LENNAERT VAN VEEN, University of Ontario Institute of Technology, GÉNIA KAWAHARA, Osaka University — Experiments, simulations and theoretical arguments lend mounting evidence for the “edge state” hypothesis on subcritical transition to shear turbulence. The hypothesis asserts that certain states of fluid motion, such as travelling waves and time-periodic flows, mediate between laminar and turbulent motion. Locally, the stable manifold of an edge state separates laminarizing from bursting flows. The global structure of the separatrix, however, is unknown. In this presentation, we show the existence of a flow homoclinic to a time-periodic edge state in plane Couette turbulence. Through classical theorems of dynamical systems theory, this implies a complex global geometry of the separating manifold. In particular, we can expect that any turbulent flow is close to the boundary, and small perturbations can cause it to re-laminarize. Also, the homoclinic flow give a preferred route from near-laminar to turbulent flow and back. We study the physical characteristics of this cycle in detail.

Monday, November 22, 2010 10:30AM - 12:40PM —
Session HÅ Turbulence Theory II — Long Beach Convention Center 101A

10:30AM HA.00001 Effect of polymer additives on bulk turbulence
  1  HENG-DONG XI, HAITAO XU, EBERHARD BODENSCHATZ, Max-Planck Institute for Dynamics and Self-Organization, Goettingen, D-37077 Germany — In recent years, there is a rising interest on the effect of polymer additives on homogeneous and isotropic turbulence. We investigate experimentally the effect of minute high-molecular-weight polymers on the bulk turbulence. The experiments are carried out in a fully developed turbulent von Karman flow between two counter-rotating baffled disks. Using the three-dimensional Lagrangian Particle Tracking technique, we follow simultaneously many tracer particles seeded in the flow, from which we extract both Eulerian and Lagrangian statistics of the turbulence. We report the results from independently varying the control parameters: the Reynolds number, the Weissenberg number, and the polymer concentration in our experiments, with the focus on the last two.

1  This work is supported by the German Science Foundation (DFG), the Max Planck Society, and the Humboldt Foundation.

3Supported by NSF.

1Supported by an NSERC Discovery grant and a Grant-in-Aid of the JSPS.
10:43AM HA.00002 de Gennes’s theory of polymer drag reduction revisited1. DONG-HYUN LEE, RAYHANEH AKHAVAN, University of Michigan — The original theory of polymer drag reduction proposed by de Gennes [1] and its re-interpretation for wall-bounded flows proposed by Sreenivasan & White [2] give predictions which are orders of magnitude off from both DNS results and available experimental data. A revised version of this theory is developed, in which the effect of the mean shear on polymer stretching is included, and the polymer is assumed to affect the dynamics of a turbulent scale when a small fraction, on the order of ~ 3%, of the turbulence kinetic energy at that scale is redirected into the elastic energy of polymer. The revised theory gives predictions in quantitative agreement with DNS and experimental results for a number of polymer drag reduction features, including the criteria for onset and of drag reduction, saturation of drag reduction, MDR, and the range of turbulent scales affected by the polymer. A complete theory of polymer drag reduction is proposed to show how this minimal exchange of energy between the polymer and turbulence can lead to the dramatic drag reductions observed with polymers.


1Supported by The Martin R. Prince Foundation & Teragrid Allocation CTSG070070.

10:56AM HA.00003 Statistics of the Energy Dissipation Rate and Local Enstrophy in Turbulent Channel Flow1. JOERG SCHUMACHER, TU Ilmenau, Germany, PETER E. HAMILTON, Naval Research Laboratory, USA, DMITRY KRASNOV, THOMAS BOECK, TU Ilmenau, Germany — Using high-resolution direct numerical simulations, the height and Reynolds number dependence of the higher-order statistics of the energy dissipation rate and local enstrophy are examined in incompressible, fully-developed turbulent channel flow. The statistics are studied at a range of wall distances, spanning the viscous sublayer to the channel flow centerline, for friction Reynolds numbers \(Re_\tau = 180\) and \(Re_\tau = 381\). The high resolution of the simulations allows dissipation and enstrophy moments up to fourth order to be calculated. These moments show a dependence on the distance from the wall, and Reynolds number effects are observed at the edge of the logarithmic layer for the enstrophy. Conditional analysis based on locations of intense vorticity is also carried out in order to determine the contribution of vortical structures to the moments of the dissipation and enstrophy. Our analysis shows that, for the simulation at the larger Reynolds number, the small-scale fluctuations of both dissipation and enstrophy become independent of distance from the wall for \(z^+ > 100\).

1We thank the DEISA Consortium, the US National Science Foundation and the Deutsche Forschungsgemeinschaft for support.

11:09AM HA.00004 On a self-sustaining process at large scale in the turbulent channel flow, YONGYUN HWANG, Laboratoire d’hydrodynamique, Ecole polytechnique, France, CARLO COSSU, Institut de Mecanique des Fluides de Toulouse (IMFT), France — The near-wall region of wall-bounded turbulent flows has been understood as the place where an independent self-sustaining cycle exists, and the associated coherent motions in this region have been rigorously described with traveling waves and/or unstable periodic orbits in the phase space. On the other hand, in the outer region, turbulent motions have often been thought to be produced from the active near-wall cycles via so called the ‘bottom-up’ process. However, recent investigations revealed that outer layer motions can experience significant non-normal amplifications. These findings suggest that self-sustaining processes could also exist at large scale. In this study, we consider a fully-developed turbulent channel at \(Re_\tau = 550\). We show that large-scale and very-large-scale motions in the outer region can sustain even when smaller-scale structures in the near-wall and the logarithmic regions are artificially quenched. The self-sustaining process is active only at the length scales larger than \(L_x \times L_z \approx 3h \times 1.5h\), in good accordance with the most energetic length scales observed in the outer region.

11:22AM HA.00005 An analytical formulation for the 1D energy spectra in equilibrium wall-bounded turbulence, YIFENG TANG, RAYHANEH AKHAVAN, University of Michigan — While a number of analytical formulations exist for the inertial and dissipation range 3D energy spectra in homogeneous, isotropic turbulence, none of these formulations can be directly applied to the near-wall region of equilibrium wall-bounded flows due to the strong anisotropy of the turbulence structure in the near-wall region. In homogeneous, isotropic turbulence, the 1D spectrum is related to the 3D spectrum through \(E(k/k_d)(z) = 2^{F(k/k_d)}\), where \(k_d\) is the Kolmogorov constant, \(F(k)\) is the dissipation range correction to the Kolmogorov spectrum, \(\varepsilon\) is the volume-averaged rate of dissipation, and \(k_d = (\varepsilon/\nu^3)^{1/5}\) is the Kolmogorov wavenumber. It is shown that an analytical formulation for the inertial and dissipation range 1D energy spectra in equilibrium wall-bounded turbulence can be obtained from \(E^{1D}(k/k_{d,\alpha})(\varepsilon_{n,\alpha})^{1/5} = 2^{F(k/k_d,\alpha)}\), where \(\varepsilon_{n}(z) = (3/16)(\partial u_{i}(\partial u_{i})_{\alpha})\) denotes the contribution of the gradients in the \(\alpha\)-direction to the total dissipation at wall-normal location \(z\). The validity of the proposed formulation is demonstrated using 1D spectra obtained from DNS databases of turbulent channel flow with \(180 < Re_\tau < 2000\).

11:35AM HA.00006 The log layer in incompressible and compressible turbulence, ROBERT RUBINSTEIN, NASA Langley Research Center — The “log law” for incompressible wall-bounded turbulence describes a self-similar flow the properties of which follow from the hypothesis that a constant stress region exists. The compressible extension is not straightforward because the density dependence requires an additional assumption. The “compressible law of the wall” of van Driest follows by requiring that the length scale be independent of density. We consider the consistency of this and alternative hypotheses with the variable density Navier-Stokes equations and derive the locality conditions imposed by the relation between the log layer and the viscous sub-layer and wake region.

11:48AM HA.00007 Quantifying the Effects of Large Scale Intermittency in Turbulence, DANIEL BLUM, GREG VOTH, Wesleyan University — We report on the effect of fluctuating energy at the largest scales on various turbulence statistics. Measurements were made in a flow between oscillating grids which contains nearly homogeneous turbulence in an 1,100 l tank which produces \(R_{\lambda} = 285\). By modulating the oscillating grid frequency we can introduce temporal variations in the injected energy which allows us to control the level of large scale intermittency. We measure the effects of this large scale intermittency by conditioning Eulerian structure functions on the large scale velocity. With constant oscillating grid frequency, the functional conditions show a clear dependence on the large scale velocity, but increasing the large scale intermittency (by increasing the frequency modulation) substantially increases this dependence. Such control allows us to quantify the effects of large scale intermittency on the various length scales of the structure functions, down to the small scales.

12:01PM HA.00008 Small-scale intermittency in anisotropic stably stratified turbulence1. SABA ALMALKIE, STEPHEN DE BRUYN KOPS, University of Massachusetts, Amherst — The statistical characteristics of small-scale turbulence in the presence of large-scale anisotropy are examined using high-resolution direct numerical simulation of stably stratified turbulence. The effects of stratification and residual anisotropy at smaller scales on turbulence intermittency is of primary interest. The scale dependency of intermittency in stratified turbulence is quantified using statistics of the locally averaged energy dissipation rate and the scaling exponent of its moments over a range of Froude numbers. The results are compared to the corresponding statistics from simulations of isotropic homogeneous turbulence with comparable numerical resolution and Reynolds numbers. The reliability of conventional surrogates of energy dissipation rate in estimating intermittency of flows dominated by large scale anisotropy is also discussed.

1Sponsored by ONR Grant N000140810236
12:14PM HA.00009 Measuring anisotropy of conditional structure functions in turbulence using real-time image compression. SUSANTHA WUJESINCHE, DANIEL BLUM, GREG VOTH, Wesleyan University — We use SO(3) decomposition to study the anisotropy of conditional structure functions in a turbulent flow between oscillating grids. The flow between two grids in a 1m x 1.5m tank achieves Reₚ=285 while having a central region that is nearly homogeneous with low anisotropy and a region near the grid with much greater inhomogeneity and anisotropy. A 3D particle tracking system with 4 high speed digital cameras records particle trajectories. We condition Eulerian velocity structure functions on the large scale velocity which reveals the effects of large scales on small scale statistics. Real-time image compression using an FPGA (Field Programmable Gate Array) makes it possible to continuously record the huge data sets necessary to decompose conditional structure functions to extract the anisotropic contributions.

12:27PM HA.00010 On Multiscale Geometrical Statistics of Anisotropic Homogeneous Turbulence. FRANK JACOBITZ, Mechanical Engineering Program, University of San Diego, San Diego, CA, USA, KAI SCHNEIDER, M2P2-CNRS & CNRS, Aix-Marseille University, Marseille, France, WOUTER BOS, LMF-CNRS, Ecole Centrale de Lyon – Université de Lyon, Ecully, France, MARIE FARGE, LMD-CNRS, Ecole Normale Superieure, Paris, France — Statistical geometrical properties of a variety of prototypical turbulent flows, including forced isotropic turbulence, sheared turbulence, rotating sheared turbulence, and rotating turbulence, are investigated in this study using results obtained from direct numerical simulations. Distributions of velocity helicity show a preference for two-dimensionalization for flows with growing turbulence and a trend to helical motion for decaying turbulence. A scale-dependent analysis shows a trend to two-dimensionalization for large scales of motion and a preference for helical motion at small scales. These results are consistent for all flows considered in this study. Joint probability distribution functions show a strong correlation of the signs of velocity helicity and vorticity helicity for all cases. This correlation supports the conjecture of Sanada (PRL 1993) that the vorticity helicity diminishes velocity helicity.

Monday, November 22, 2010 10:30AM - 12:40PM — Session HB Turbulent Boundary Layers V Long Beach Convention Center 101B

10:30AM HB.00001 Investigation of Turbulent Wall Pressure Fluctuations over a wide Reynolds Number Range of Turbulent Pipe Flows. JAN COOK, Cornell Univ, WILLIAM KEITH, ALIA FOLEY, KIMBERLY GIPOLLA, NAVSEA Newport. — The flow noise induced by turbulent wall pressure fluctuations under a moderate to high Reynolds number turbulent boundary layer constitutes a primary limitation on acoustic array performance. The widest and highest ranges of the Reynolds numbers of interest are most easily achieved in pipe flow configurations. It is therefore of interest to investigate the turbulent boundary layers which are generated on a pipe wall to assess if this class of flows can sufficiently replicate the turbulent boundary layers to which an acoustic array is exposed. Turbulent wall pressure fluctuations were measured under the aqueous turbulent boundary layer in a 3.5-inch diameter circular pipe. A linear array of small diameter wall pressure sensors flush mounted in the pipe wall recorded wall pressure fluctuations for a range of centerline flow velocities. These data were used to perform spectral analysis of the turbulent energy in the flow. Reynolds number effects were explored and comparisons were made with wall pressure fluctuations under flat plate turbulent boundary layers.

1Supported by ONR NREIP

10:43AM HB.00002 Mean profile of a high-Reynolds-number smooth-flat-plate turbulent boundary layer. DAVID R. DOWLING, Univ. of Mich., GHANEM F. OWEIS, Am. Univ. of Beirut, ERIC S. WINKEL, Design Research Engineering, JAMES M. CUTBIRTH, Mainstream Engineering Corp., STEVEN L. CECCIO, MARC PERLIN, Univ. of Mich. — Although smooth-flat-plate turbulent boundary layers (TBLs) have been studied for nearly a century, measurements at Reynolds numbers typical of marine & aerospace transportation systems are scarce. Experimental results at momentum-thickness Reynolds numbers (Re) up to 150,000 from the US Navy W.B. Morgan Large Cavitation Channel using a polished 12.9-m-long flat-plate test model at water flow speeds up to 20 m/s are presented. Mean velocity profiles were measured 10.7 m from the leading edge of the model over a wall-normal range from less than one wall unit to more than twice the nominal boundary layer thickness using particle-tracking and laser-Doppler velocimetry. Static pressure and average skin-friction were measured independently. A mild favorable pressure gradient led to a flow speed increase of 2.5% over the test surface. The measurements span a factor of three in Re and were fitted to within experimental uncertainty using one set of constants and modern empirical inner- and outer-profiles based on traditional TBL asymptotics. The fitted profiles satisfy the von-Karman momentum integral to within 1%, and show distinct differences from equivalent zero pressure gradient results. [Supported by DARPA & ONR]

10:56AM HB.00003 Turbulence in Favorable pressure gradient (FPG) boundary layers. PRANAV JOSHI, JOSEPH KATZ, Johns Hopkins University — Our objective is to study the effect of favorable pressure gradient on near wall structures in a sink flow turbulent boundary layer over a smooth wall. 2D PIV measurements have been performed upstream of and within the region of constant acceleration parameter, K=\nu U_0/U_{\infty}^{2/3}, of 0.575\times10^{-6}. In the initial range, where K increases to its asymptotic value, all the Reynolds stresses and skin friction coefficient, C_f, decay. In the region of constant K, the stresses continue to decay in the outer layer, but C_f and all the Reynolds stress components increase close to the wall (C_f/\delta=0.2). The stresses collapse when scaled with the local freestream velocity, U_0(x). TKE production and wall normal transport of turbulence also scale with U_0(x)^3/\delta(x) close to the wall. PIV data obtained in wall-parallel planes show the expected low speed streaks (LSS) bounded by large structures in the zero pressure gradient range. Narrower LSS persist also in the constant K area, but the signatures of large structures diminish. In both regions, small-scale structures, with signatures suggesting inclined quasi-streamwise vortex pairs, appear predominantly in the LSS areas, suggesting that they are preferred sites of turbulence production.

1Supported by NSF (Grant No.0932941)

11:09AM HB.00004 Roughness Signature in the Outer Layer of a Turbulent Boundary Layer. JIARONG HONG, JOSEPH KATZ, Johns Hopkins University, MICHAEL SCHULTZ, United States Naval Academy — Roughness signature, consisting of bumps (slope flattening) in energy spectra at roughness scale wavenumbers, have been observed in the outer-layer high resolution PIV data obtained in a turbulent channel flow over 3D surfaces. The measurements cover the entire well-characterized channel flow with \delta/k=50 (k is roughness height) and k_z=90-150. For the present Reynolds numbers, Re_\tau=3520-5360, these spectral bumps fall in 10-30 times the local Kolmogorov scale. Instantaneous realizations, swirling strength based linear stochastic estimation, and bandpass-filtered velocity maps indicate that this phenomenon is a result of rapid entrainment of eddies generated near the wall by large scale, outer-layer structures. This process floods the boundary layers with eddies of 1-3 times the roughness height, in addition to those generated by local production. Consequently, the energy and shear spectra show an excessive amount of turbulence energy in the outer layer. On-going time-resolved measurement focus in this interaction between inner and outer layer structures. Thus, although the means flow and second order moment statistics satisfy Townsend’s hypothesis, the small scale turbulence maintains the roughness signature.

1Supported by ONR.
11:22AM HB.00005 Turbulence Structure in Oscillating Channel Flow. SEAN KEARNEY, JESSE ROBERTS, Sandia National Laboratories, JOSEPH GAILANI, US Army Corps of Engineers — The structure of turbulence in an oscillating channel flow with near-sinusoidal fluctuations in bulk velocity is investigated. Phase-locked particle-image velocimetry data in the streamwise-wall-normal plane are interrogated to reveal the phase-modulation of two-point velocity correlation functions and of linear stochastic estimates of the velocity fluctuation field given the presence of a vortex in the logarithmic region of the boundary layer. The results reveal the periodic modulation of turbulence structure between large-scale residual disturbances, relaminarization during periods of strong acceleration, and a quasi-steady flow with evidence of hairpin vortices which is established late in the acceleration phase and persists through much of the deceleration period.

11:35AM HB.00006 Analysis of Velocity Measurements on a Towed, Flexible Cylinder*. BRIAN AMARAL, URI KIMBERLY CIOPOLLA, NAVŠEA Newport — High resolution stereo-PIV measurements were made on a long, flexible cylinder towed in the David Taylor Model Basin. The experiments were performed from 12 to 30 kts to generate Reynolds numbers based on momentum thickness greater than one million. The cylinders (130 m long, 38 mm diameter) were approximately neutrally buoyant and towed through a stationary laser sheet oriented perpendicular to the tow direction to obtain 3D velocity fields. The objective of the study was to quantify the mean and fluctuating velocity fields in the turbulent boundary layer on an experimental towed array (flexible cylinder) where boundary layer thickness is much greater than the cylinder radius. Algorithms for image pre-processing and filtering were applied to enhance the instantaneous images and mask the cylinder and its shadow as they move in the 80 cm x 85 cm field of view. Relevant boundary layer parameters were determined as a function of streamwise position and Reynolds number. Initial results indicate that the velocity defect law provides the best collapse of the data in the outer region of the boundary layer, while the log law relation is effective very close to the surface of the cylinder.

*Supported by ONR NREIP

11:48AM HB.00007 Effect of wall-mounted cylinders on a turbulent boundary layer: hot wire measurements*. CECELIA ORTZ-DUEÑAS, Institute for Mathematics and its Applications, University of Minnesota, MITCHELL RYAN, ELLEN LONGMIRE, Department of Aerospace Engineering and Mechanics, University of Minnesota — Wall-mounted cylinders with height-to-diameter ratio H/D = 2 and large enough to protrude into the logarithmic region, H+ = 200, are used to alter a turbulent boundary layer with Re∞=1150 in an attempt to affect the organization of the coherent vortical structures. Hot-wire measurements, including velocity profiles and frequency spectra, were acquired downstream of a single cylinder and spanwise arrays of cylinders. The single cylinder yields a momentum deficit that extended from z+=20 to 200, and a redistribution of the streamwise rms velocity towards the half cylinder height with a corresponding increase in the power spectral density over a broad frequency range. Cylinder arrays with 3D spanwise spacing yielded significant wake interactions. The largest mean streamwise velocity deficits and rms values occurred in the log region at mid-span between cylinders. More detail on the effect of cylinder spacing will be provided in the talk. The results suggest that turbulence within the boundary layer leads to broader spanwise interactions than those occurring in wakes of cylinder arrays in uniform cross flow.

*Supported by NSF (CBET-0933341) and University of Minnesota IMA.

12:01PM HB.00008 Effect of wall-mounted cylinders on a turbulent boundary layer: V3V measurements*. MITCHLYR RYAN, CECELIA ORTZ-DUEÑAS, ELLEN LONGMIRE, Department of Aerospace Engineering and Mechanics, University of Minnesota, DAN TROOLIN, TSI Incorporated — Volumetric 3-Component Velocimetry (V3V) was used to examine the flow structure downstream of arrays of wall-mounted-cylinders in a turbulent boundary layer with Re∞=2460. The cylinders, which had height-to-diameter ratio H/D = 4 and H+ = 455, extended through the logarithmic region. Measurements were acquired in fields that extended over a range 16 to 34 cylinder-diameters downstream of spanwise arrays of cylinders with a spacing of four and eight cylinder diameters (0.2δ and 0.4δ). The cylinder array with 4D spacing yielded significant wake interactions: the streamwise velocity deficit was greater at the mid-spanning than directly behind a cylinder; the distinction between the downstream regions (behind a cylinder) and the upwash regions (at the mid-spacing) diminishes with increasing downstream distance; and the rms velocity in all components is highest at the half-cylinder height. These effects occur to a much lesser degree in the case of the array with 8D spacing. Details on parametric effects as well as the instantaneous three-dimensional structure will be provided in the talk.

*Supported by NSF (CBET-0933341).

12:14PM HB.00009 The Drag of 2D Single-Roughness Elements Immersed in Turbulent Boundary Layers*. BOON TUAN TEE, TIMOTHY NICKELS, University of Cambridge — Most of drag studies on flow behind single protuberances in laminar boundary layers focusing mainly with transition, with few studies as regards to turbulent boundary layers. The primary aim of this work was to determine the drag of these protuberances since they exist in practical flows. The experiment was conducted in the turbulence water tunnel research facility at Cambridge University Engineering Department. Measurements were taken using 2D-PIV on 3 types of rough wall configuration involving 2D rough bar with the triangular, circular and semi-circular shapes. The total drag for these 3 types of single protuberances was calculated by considering the undisturbed boundary layer upstream and downstream of the boundary layer. The result reveals that the drag is dependent on the area facing the flow with the triangular shape creating the most drag and the semi-circular the least drag.

*Supported by NSF CBET-0933341.

12:27PM HB.00010 Turbulence characteristics around a staggered wind farm configuration: A wind tunnel study. LEONARDO CHAMORRO, ROGER ARNDT, FOTIS SOTIROPOULOS, University of Minnesota — Turbulent flow around a wind farm is characterized by the coexistence and superposition of multiple wind turbine wakes. The understanding of the momentum transport and velocity fluctuations at different locations in the wind farm is essential to improve energy production and the structural stability of the different turbines. In this study, a staggered model wind farm was placed in the boundary layer wind tunnel of the Saint Anthony Falls Laboratory at the University of Minnesota. The staggered wind farm consisted of 10 rows in the streamwise direction by 2–3 columns. A cross-wire anemometer was used to obtain high-resolution measurements of 2 velocity components (streamwise and vertical) inside and above the model staggered wind farm. Full characterization of the turbulent flow was obtained at a vertical plane parallel to the flow direction through the entire wind farm and at 4 spanwise vertical planes (located at 5 rotor diameter behind the 4th, 6th, 8th and the 10th row). Special emphasis is placed on the description of the enhancement of the turbulence levels in the wind farm as a function the number of rows of the wind farm as well as the growth of the internal boundary layer induced by the wind farm. The results are being used to develop new parameterizations of wind turbines for high-resolution and large-scale numerical models.

Monday, November 22, 2010 10:30AM - 12:27PM
Session H6 General Fluid Dynamics II Long Beach Convention Center 102A
10:30AM HC.00001 A regularization by stratification of the Stokes flow divergences by translating spheres. ROBERTO CAMASSA, CLAUDIA FALCON, University of North Carolina, JOYCE LIN, University of Utah, RICHARD MCLAUGHLIN, ANNA MILLER, KATHRYN VALCHAR, University of North Carolina, UNC NSF RTG FLUIDS GROUP COLLABORATION — Stokes flow solutions of fluid motions in the presence of a moving sphere notoriously suffer unphysical divergences in quantities such as the dragged volume of fluid which have been traditionally regularized by far-field Oseen inertial corrections. This talk will consider an alternative regularization mechanism related to the presence of stable stratification under gravity in the fluid. A first principle theory will be outlined and results with falling spheres in sharp stratifications will illustrate the mechanism.

10:43AM HC.00002 Super free fall for a container composed of diverging flat plates. A. MEDINA, A. TORRES, S. PERALTA, SEPI ESIME-Unidad Azcapotzalco IPN, P.D. WEIDMAN, University of Colorado at Boulder — We have analyzed experimentally and theoretically the characteristics of the upper free surface of a liquid column released from rest in a vertical container whose cross-section opens slowly in the downward direction. In distinction with the work of Villermaux and Pomeau (2010) for a conical container, we consider a container composed of slightly inclined flat surfaces. At small times for which viscous effects can be neglected, the free surface moves downward with an acceleration larger than gravity. The existence of a nipple centered on the upper free surface with amplitude an increasing function of time is observed. A one-dimensional model of the initial acceleration for flat, slightly expanding walls reproduces the observed super free fall experiments fairly well. Details of the nipple development will be presented.

10:56AM HC.00003 Ratcheting Fluid using Geometric Anisotropy. BENJAMIN THIRIA, PMMH-ESPCI, France, JUN ZHANG, Physics Department and Courant Institute, NYU, USA — We discuss a new type of pump that can effectively transport fluids using vibrational motion imposed onto shapes with anisotropy. In our experiment, two asymmetric, sawtooth-like structures are placed facing each other and form a fluid channel. This channel is then flooded with open and closed periodically. Under geometric, reciprocal motion, fluid fills in the gap during the expansion phase of the channel and is then forced out during contraction. Since the fluid experiences different impendence when it takes different directions in the gap, the stagnation point that separates flows of two directions changes within one driving period. As a result, fluid is transported or pumped from one end of the gap to the other. This ratcheting effect of fluid is demonstrated through our measurements and its working principle is discussed in some detail. We also discuss the potential applications of this vibratory fluid pump.

11:09AM HC.00004 Swelling-Induced Dynamic Responses of Soft Materials: Bending, Buckling, and Twisting. TARUN SINHA, DOUGLAS HOLMES, MATTHIEU ROCHE, Princeton University, ARNAUD SAINT-JALMES, EQUIPE Biophysique, Institut de Physique de Rennes, France, GEORGE MAURDEV, CSIRO Melbourne, AU, HOWARD STONE, Princeton University — Soft materials (e.g. tissue, gels) undergo volume changes and instabilities when subjected to external stimuli. We present the dynamic instabilities that occur by straining an elastomer anisotropically. We examine how thin elastic plates can undergo rapid bending and buckling instabilities after exposure of the crosslinked, elastic network to a favorable solvent that causes it to swell. The shape of the swollen material is determined by the minimization of the system’s bending energy in conjunction with any external forces, or constraints on the geometry; here we focus on dynamics. An unconstrained beam bends along its length, while a circular disc bends and buckles with multiple curvatures that rotate azimuthally around the disc. Theoretical interpretations motivated by the complementary thermal expansion problem of transient shape changes triggered by time-dependent heating will be presented. Developing a quantitative understanding of this phenomena will not only further explain the dynamics of morphogenesis in growing soft tissues, but also will lead to the creation of advanced elastic materials that can adapt to stimuli to change shape, and possibly direct and control fluid flow.

11:22AM HC.00005 Study of Thermo-Acoustic Instabilities in a Rijke Tube Without and With Porous Inert Medium. CODY OSMER, Shelton State Community College, AJAY AGRAWAL, University of Alabama — Porous inert medium has been used in the past to reduce combustion noise in atmospheric pressure systems. It is envisioned that this same approach could be used to mitigate combustion instabilities encountered in gas turbines using lean premixed combustion concept. In this study, a simple Rijke tube is used to investigate the effect of porous inert media on thermo-acoustic instabilities. The Rijke tube set up utilizes a simple tube with a localized heat source. Sound pressure level measurements are taken at the tube exit to determine the sound power spectra in decibels. Such measurements were taken without and with porous medium inserted within the Rijke tube. The porous media is a 2.54 cm thick ceramic disk with 10 to 20 pores per inch. Results show that the Rijke tube behavior without the porous media follows the trends observed by previous investigators. The porous media was effective in reducing the sound pressure level, and thus, it offers the potential to mitigate thermo-acoustic instabilities through proper geometric design.

11:35AM HC.00006 Effect of Leading Edge Tubercles on Marine Tidal Turbine Blades. MARK MURRAY, TIMOTHY GRUBER, DAVID FREDRIKSSON, U.S. Naval Academy — This project investigated the impact that the addition of leading edge protuberances (tubercles) have on the effectiveness of marine tidal turbine blades, especially at lower flow speeds. The addition of leading edge tubercles to lifting foils has been shown, in previous research, to delay the onset of stall without significant hydrodynamic costs. The experimental results obtained utilizing three different blade designs (baseline and two tubercle modified) are compared. All blades were designed in SolidWorks and manufactured utilizing rapid prototype techniques. All tests were conducted in the 120 ft tow tank at the U.S. Naval Academy using a specifically designed experimental apparatus. Results for power coefficients are presented for a range of tip speed ratios. Cut-in velocity is also compared between the blade designs. For all test criteria, the tubercle modified blades significantly outperformed the smooth leading edge baseline design blades.

11:48AM HC.00007 Evaporation and impact of water droplet on superhydrophobic surfaces. PEICHUN TSAI, MAURICE HENDRIX, REMCO DIJKSTRA, ROB LAMMERTINK, MATTHIAS WESSLING, DETLEF LOHSE — We examine both quasi-static and dynamic effects of water droplets upon hydrophobic microstructured surfaces, which possesses a high contact angle ~ 150° for the droplet size of ~ 1 mm in radius. First, a milli-meter sized water droplet sitting on microstructures under a natural evaporation can undergoes a transition from a heterogeneous (Cassie-Baxter) to a homogenous (Wenzel) wetting state, when the droplet size is reduced to about a couple hundred microns. The contact angle changes during the evaporation. With the evolution of the contact angle, a model based on global surface energies was developed to predict the transition points, which agree well with the experimental data. Secondly, water droplet impinging on the superhydrophobic surface can completely rebound off the surface when the impacting kinetic energy is comparable with the surface energy. As an increase of kinetic energy is about a few hundred times larger than the surface tension, a splash—emitting satellite droplets—occurs during the advancing phase of the lamella. We will discuss the influence of the geometric patterns on the splash.

1Work performed under REU site sponsored by NSF grant EEC 0754117.
and the inertia of the returning flow was examined for a closed loop system. The effects on the average flow rate of varying the pinching frequency and the pinching position were investigated. The interaction between the wave dynamics and the elastic tube at a position that is asymmetric with respect to its ends. Two valveless pumps are chosen, a single valveless pump and a double valveless pump. Staggered Cartesian grid system. The fluid motion defined on an Eulerian grid and the structure motion defined on a moving Lagrangian grid are independently using the variational derivative of the deformation energy. Our method is based on an efficient Navier-Stokes solver that uses the fractional step method and a non-iterative coupling scheme is developed for the simulation of a viscous flow interacting with multiple bodies. Several cases are examined to demonstrate the accuracy, simplicity and efficiency of the new method.

A sharp interface immersed boundary method is presented for the simple and efficient simulation of fluid-structure interaction with complex three-dimensional rigid bodies. The previous formulation by Yang and Balaras (An Embedded-Boundary Formulation for Large-Eddy Simulation of Turbulent Flows Interacting with Moving Boundaries, J. Comput. Phys. 215 (2006) 12-40) is greatly simplified without sacrificing the overall accuracy. In addition, a novel, highly efficient non-iterative coupling scheme is developed for the simulation of a viscous flow interacting with multiple bodies. Several cases are examined to demonstrate the accuracy, simplicity and efficiency of the new method.

1Supported in part by U.S. Dept. of Energy and Michelin Americas Research & Development Corp.

**Monday, November 22, 2010 10:30AM - 12:27PM**

Session HD CFD II: Immersed Boundary Methods and Fluid-Structure Interaction

**10:30AM HD.00001 ABSTRACT WITHDRAWN —**

**10:43AM HD.00002 Curvilinear Immersed Boundary Method for Simulating Sediment Transport and Scour in Open Channel Flows**1, ALI KHOSRONEJAD, SEOKKOO KANG, IMAN BORAZJANI, FOTIS SOTIROPOULOS, St. Anthony Falls Lab, University of Minnesota — The fluid-structure interaction curvilinear immersed boundary numerical method of Borazjani et al. (J. Comput. Phys. 2008) is extended to simulate coupled flow and sediment transport phenomena. The method is inherently suited for carrying out the breakup length of the liquid sheet showed a dependence on Weber number proportional to We$^{-1/6}$, for Weber numbers of 2700, 10900 and 24400. A technique to identify and size water droplets was developed and the distribution of droplet sizes was determined as a function of Weber number. At We = 2700, droplet sizes between 80 and 9000µm were detected, with a mean diameter near 800µm. Both the range of droplet sizes and the mean diameter were found to decrease with increasing Weber number as (approximately) We$^{-1/2}$. Correlation Image Velocimetry (CIV) was used to estimate the distribution of droplet velocities as a function of droplet size. The spread of droplet velocities about the tire peripheral speed is strongly correlated with droplet size. The spread can be estimated by a simple physical model incorporating rigid droplets subject to gravity and drag.

1Supported by Michelin Americas R and D.

**10:56AM HD.00003 A Simple and Efficient Sharp Interface Immersed Boundary Method for Fluid-Structure Interaction with Complex Rigid Bodies**1, JIANMING YANG, FREDERICK STERN, University of Iowa — A sharp interface immersed boundary method is presented for the simple and efficient simulation of fluid-structure interaction with complex three-dimensional rigid bodies. The previous formulation by Yang and Balaras (An Embedded-Boundary Formulation for Large-Eddy Simulation of Turbulent Flows Interacting with Moving Boundaries, J. Comput. Phys. 215 (2006) 12-40) is greatly simplified without sacrificing the overall accuracy. In addition, a novel, highly efficient non-iterative coupling scheme is developed for the simulation of a viscous flow interacting with multiple bodies. Several cases are examined to demonstrate the accuracy, simplicity and efficiency of the new method.

1Supported by the US Office of Naval Research grants EAR-0120914 and EAR-0738726 and the University of Minnesota Supercomputing Institute.

**11:09AM HD.00004 Three-dimensional simulation of a valveless pump**1, SOO JAI SHIN, HYUNG JIN SUNG, Department of Mechanical Engineering, KAIST — An immersed boundary (IB) method for simulating a three-dimensional valveless pump is described. The valveless pump is treated as an elastic tube connected at its ends to a rigid tube. The governing equation for the motion of the elastic tube is derived by using the variational derivative of the deformation energy. Our method is based on an efficient Navier-Stokes solver that uses the fractional step method and a staggered Cartesian grid system. The fluid motion defined on an Eulerian grid and the structure motion defined on a moving Lagrangian grid are independently solved, and their interaction is formulated by using momentum forcing. A net flow is generated inside the valveless pump through the periodic pinching of the elastic tube at a position that is asymmetric with respect to its ends. Two valveless pumps are chosen, a single valveless pump and a double valveless pump. The effects on the average flow rate of varying the pinching frequency and the pinching position were investigated. The interaction between the wave dynamics and the inertia of the returning flow was examined for a closed loop system.

1This work was supported by the Creative Research Initiatives and World Class University programs of MEST/NRF.
11:22AM HD.00005 Simulations of the Motion of Arbitrarily Shaped Fibers in a Linear Shear Flow\textsuperscript{1}, ANDRIY ROSHCHENKO, Department of Mathematical and Statistical Sciences, University of Alberta, WARREN FINLAY, Department of Mechanical Engineering, University of Alberta, PETER MINEV, Department of Mathematical and Statistical Sciences, University of Alberta — Fibrous airborne particles cause severe adverse health effects when inhaled and deposited in human lungs. For this reason, fiber deposition in the lungs has been studied by numerous authors. However, a complete mechanistic model of fiber dynamics in the lungs has not yet been presented. One of the problems yet to be addressed involves the dynamics of arbitrarily shaped fibers in the lungs. Here, a two-grid fictitious domain method was used for direct simulations of arbitrarily shaped high aspect ratio fibers in a linear shear flow, including an improved microscale grid resolution scheme and a Lagrangian-Eulerian approach whereby we transform the equations from a laboratory coordinate system to one fixed with the microgrid. Our simulations showed the expected Jeffery orbits for straight, symmetric fibers.

1\textsuperscript{1}Supported by NSERC Discovery Grant.

11:35AM HD.00006 Fluid-Solid Interactive Methodology for Prognosis of Passenger Jet Structural Damage in Water Crash Landing, JAVID BAYANDOR, Virginia Tech — Today, crashworthiness studies constitute a major part of modern aerospace design and certification processes. Of important consideration is the assessment of structural damage tolerance in terms of the extent of progressive damage and failure caused by aircraft emergency ditching on soft terrain or on water. Although a certification requirement, full scale crash landings are rarely tested using fully functional prototypes due to their high associated costs. This constraint makes it difficult for all crashworthy features of the design to be identified and fine-tuned before the commencement of the manufacturing phase. The current study presents aspects of a numerical methodology that can drastically subside the dependency of the certification assessments to full scale field trials. Interactive, fully nonlinear, solid-structure and fluid-structure analyses have been proposed using coupled Lagrangian-Eulerian and independent meshless Lagrangian approaches that run on a combined finite element-computational fluid dynamics platform. Detailed analysis of a key landing scenario pertaining to a large passenger jet will be provided to determine the relevance and accuracy of the proposed method. The work further identifies state-of-the-art computational approaches for modeling fluid-solid interactive systems that can help improve aircraft structural responses to soft impact and water ditching.

11:48AM HD.00007 A discrete-forcing immersed boundary method for the fluid-structure interaction of an elastic slender body\textsuperscript{1}, INJAE LEE, HAECHEON CHOI, Seoul National University — In the present study, an immersed boundary method for the simulation of flow around an elastic slender body is developed. The present method is based on the discrete forcing method by Kim et al. (J. Comput. Phys., 2001) and is fully coupled with the elastic slender body motion. The incompressible Navier-Stokes equations are solved in an Eulerian coordinate and the elastic slender body motion is described in a Lagrangian coordinate, respectively. The elastic slender body is assumed as a thin flexible beam and is segmented by finite blocks. Each block is then moved by the external and internal forces such as the hydrodynamic, tension, bending, and buoyancy forces. We simulate several flow problems and the results agree very well with those from previous studies. Moreover, the present method does not impose any severe limitation on the size of computational time step due to the numerical stability.

1\textsuperscript{1}Supported by the NRL and WCU Programs, KRF, MEST, Korea.

12:01PM HD.00008 Effects of mass ratio to flexible flapping-wing propulsion\textsuperscript{1}, MIN XU, MINGJUN WEI, TAO YANG, THOMAS BURTON, New Mexico State University — In our previous work, we used a strong-coupling approach to simulate highly flexible wings interacting with surrounding fluid flows. However, there was a strong assumption: the wing structure has the same density as the surrounding fluid. Though this assumption has greatly simplified the formulation and worked well in most of our previous studies, it made impossible to consider the effects of mass ratio between the structure and fluid. In this study, we introduced another body force term to represent the density difference and also modified the formulation so that almost no extra cost was added in order to consider the mass ratio effects. Using the new algorithm, we found an interesting nonlinear response of the trailing-edge frequency to the active plunging frequency at the leading edge when certain flapping frequency and mass ratio were chosen.

1\textsuperscript{1}Supported by ARL through Army High Performance Computing Research Center.

12:14PM HD.00009 A fully-coupled approach to simulate three-dimensional flexible flapping wings\textsuperscript{1}, TAO YANG, MINGJUN WEI, New Mexico State University — The algorithm in this study is based on a combined Eulerian description of both fluid flow and solid structure which then can be solved in a monolithic manner. Thus, the algorithm is especially suitable to solve fluid-structure interaction problems involving large and nonlinear deformation. In fact, we have successfully applied the same approach to our previous study of two-dimensional pitching-and-plunging problems and found many unique features from the passive pitching introduced by wing flexibility. With the current non-trivial extension of the algorithm to three-dimensional configuration, we can eventually reveal the complex vortex and structural dynamics behind the amazing performance of nature’s fliers such as hummingbirds.

1\textsuperscript{1}Supported by ARL through Army High Performance Computing Research Center.
different approaches are used to illustrate its influence: a local stability analysis of a family of wakes introduced by Monkewitz (1988).

The goal of this communication is to understand how the presence of a finite shear layer thickness in a confined wake modifies the stability properties of the flow. Two-dimensional direct numerical simulations and a global linear stability analysis. The direct numerical simulations and the global stability results accurately capture the evolution of vortex shedding and the transition regions to turbulence as well as flow separations along the sidewall. The formation of a vortex street is generally considered to be the result of a critical Reynolds number for the steady bifurcation, \( \Re \), increases monotonically as \( D/D_c \) increases, reaching an asymptotic value, that depends on \( D/D_c \) at \( h/D \approx 0.7 \). On the other hand, for a fixed value of \( h/D \), \( \Re \) exhibits a maximum at \( D/D_c \approx 0.8 \). Similar behavior has been observed experimentally and numerically for the second, oscillatory bifurcation, and its associated critical Reynolds number, \( \Re_{osc} \).

We extend our previous research on the instability properties of the laminar incompressible flow around a cylindrical body with a rounded nose and length-to-diameter ratio \( L/D = 2 \), at zero angle of attack, by analyzing the effects of a cylindrical base cavity of length \( h \) and diameter \( D_c \). We combine experiments, three-dimensional direct numerical simulations and a global linear stability analysis. The direct numerical simulations and the global stability results accurately predict the stabilizing effect of the cavity on the stationary, three-dimensional bifurcation in the wake as \( h/D \) increases. In fact, it is shown that, for a given value of \( D/D_c \), the critical Reynolds number for the steady bifurcation, \( \Re_{st} \), increases monotonically as \( h/D \) increases, reaching an asymptotic value, that depends on \( D/D_c \) at \( h/D \approx 0.7 \). On the other hand, for a fixed value of \( h/D \), \( \Re_{st} \) exhibits a maximum at \( D/D_c \approx 0.8 \). Similar behavior has been observed experimentally and numerically for the second, oscillatory bifurcation, and its associated critical Reynolds number, \( \Re_{osc} \).

Simultaneously, some numerical simulations were concerned with cylindrical bodies of finite thickness and showed that the wakes of such bodies present a different scenario from that of a flat disk and that of a sphere (which is widely known and accepted). A systematic study covering the whole range of cylinders of aspect ratio (diameter/thickness) between one and infinity as well as a study concerning oblate spheroids which establishes the link between the wake of a sphere and that of an infinitely flat disk, which until now was missing, will be the topic of this communication. The state diagram obtained for oblate spheroids illustrating the transition between the scenario of a sphere wake and that of a flat disk will be presented and discussed.

The interaction between KH vortical structures in the separating shear layer and Karman vortex shedding in the near wake will be discussed based on both visualisations and frequency analysis. In particular, the dependency with Reynolds number of the ratio from the shear layer frequency to the fundamental Karman frequency by Bloor (1964) will be investigated for the square cylinder. The controversial resulting square root law discussed by Rajagopalan and Antonia (2005) will be focused for the square cylinder case as well.

1. This work was financially supported by the Swedish Research Council and the Göran Gustafsson Foundation. Travel grant by the Bengt Ingeström Foundation is also acknowledged.

11:22AM HE.00005 Stability effects of a base cavity on the wake of axisymmetric bluff bodies. MARCIN CHRUST, Institute of Fluid and Solid Mechanics, University of Strasbourg, GILLES BOUCHET, Institute of Fluid and Solid Mechanics, University of Strasbourg/CNRS, JAN DUSEK, Institute of Fluid and Solid Mechanics, University of Strasbourg — Recently, the wake of a flat disk has regained the interest of researchers simultaneously, some numerical simulations were concerned with cylindrical bodies of finite thickness and showed that the wakes of such bodies present a different scenario from that of a flat disk and that of a sphere (which is widely known and accepted). A systematic study covering the whole range of cylinders of aspect ratio (diameter/thickness) between one and infinity as well as a study concerning oblate spheroids which establishes the link between the wake of a sphere and that of an infinitely flat disk, which until now was missing, will be the topic of this communication. The state diagram obtained for oblate spheroids illustrating the transition between the scenario of a sphere wake and that of a flat disk will be presented and discussed.


11:35AM HE.00006 The influence of shear layer thickness in the stability of confined 2D wakes. ERIC SERRE, M2P2 UMR6181 CNRS Aix-Marseille Université, Marseille, MATTHIEU MINGUEZ, Seal Engineering, Centre Atria, Nimes, CHRISTOPHE BRUN, LEGI, UMR CNRS 5519, Université Joseph Fourier, Grenoble, RICHARD PASQUETTI, Laboratoire J.-A. Dieudonné, UMR CNRS 6621, Université de Nice-Sophia Antipolis, Nice — The flow that develops behind a cylinder is very complex because it is fully three-dimensional, unsteady, including transition regions to turbulence as well as flow separations along the sidewall. The formation of a vortex street is generally considered to be the result of a coupling between Kelvin-Helmholtz instabilities within the separated shear layers and the Karman instability in the near wake. In the present paper we propose a joint experimental / numerical study in order to investigate the flow features in the near wall region of a square cylinder at \( Re = 21400 \) (ERCOFTAC benchmark). The interaction between KH vortical structures in the separating shear layer and Karman vortex shedding in the near wake will be discussed based on both visualisations and frequency analysis. In particular, the dependency with Reynolds number of the ratio from the shear layer frequency to the fundamental Karman frequency by Bloor (1964) will be investigated for the square cylinder. The controversial resulting square root law discussed by Rajagopalan and Antonia (2005) will be focused for the square cylinder case as well.
Low-Reynolds numbers vortex-induced vibrations by means of asymptotic methods, JEAN-MARC CHOMAZ, LadHyX-Ecole Polytechnique - CNRS, PHILIPPE MELIGA, Ecole Polytechnique Federale de Lausanne - Lab. of Fluid Mechanics and Instabilities — The present work investigates the dynamics of a spring-mounted circular cylinder by focusing on the amount of energy that can be extracted from the flow when an appropriate forcing is applied, which is of practical interest when vortex-induced vibrations are thought to be used for energy production. The analysis relies on an asymptotic model developed at low Reynolds numbers, herein extended to encompass the effect of the forcing. In practice, we consider the case of an actuator prescribing a small, periodical blowing and suction velocity at the cylinder wall. We vary the structural damping and natural frequency of the cylinder, and characterize the dynamics of the forced, nonlinear limit cycles. We then evidence that the magnitude of extracted energy can be maximized using the framework of the optimal control theory, which relies on an iterative algorithm based on the repeated computation of adjoint fields. A physical interpretation for the optimal control will also be proposed, in terms of the cylinder displacement, velocity and acceleration.

Energy extraction from a low Reynolds-number-flow using vortex-induced vibrations and optimal control, PHILIPPE MELIGA, Ecole Polytechnique Federale de Lausanne - Lab. of Fluid Mechanics and Instabilities, JEAN-MARC CHOMAZ, LadHyX-Ecole Polytechnique - CNRS — The present work investigates the dynamics of a spring-mounted circular cylinder by focusing on the amount of energy that can be extracted from the flow when an appropriate forcing is applied, which is of practical interest when vortex-induced vibrations are thought to be used for energy production. The analysis relies on an asymptotic model developed at low Reynolds numbers, herein extended to encompass the effect of the forcing. In practice, we consider the case of an actuator prescribing a small, periodical blowing and suction velocity at the cylinder wall. We vary the structural damping and natural frequency of the cylinder, and characterize the dynamics of the forced, nonlinear limit cycles. We then evidence that the magnitude of extracted energy can be maximized using the framework of the optimal control theory, which relies on an iterative algorithm based on the repeated computation of adjoint fields. A physical interpretation for the optimal control will also be proposed, in terms of the cylinder displacement, velocity and acceleration.

10:30AM HE.00001 Experimental study of the flow over a backward-facing rounded ramp, THOMAS DURIEZ, JEAN-LUC AIDER, JOSE EDUARDO WESFREID, Laboratoire PMMH UMR 7636 CNRS ESPCI ParisTech - Universite Pierre et Marie Curie - Universite Paris Diderot, INSTABILITY CONTROL OF TURBULENCE TRENDS — The backward-facing rounded ramp (BFR) is a very simple geometry leading to boundary layer separation, close to the backward facing step (BFS) flow. The main difference between the BFS flow is that the separation location depends on the incoming flow while it is fixed to the step edge for the BFS flow. Despite the simplicity of the geometry, the flow is complex and the transition process still has to be investigated. In this study we investigate the BFR flow using time-resolved PIV. For Reynolds number ranging between 300 and 12 000 we first study the time averaged properties such as the positions of the separation and reattachment, the recirculation length and the shear layer thickness. The time resolution also gives access to the characteristic frequencies of the time-dependent flow. An appropriate Fourier filtering of the PIV data allows us to focus on high frequency peaks in the global spectrum, allowing an investigation of each mode in order to extract its wavelength, phase velocity, and spatial distribution. We then sort the spectral content and relate the main frequencies to the most amplified Kelvin-Helmholtz instability mode and its harmonics, the vortex pairing, the low frequency recirculation bubble oscillation and the interactions between all these phenomena.

10:43AM HE.00002 Forcing the shear layer of a backward-facing step flow using DBD plasma actuator, JEAN-LUC AIDER, THOMAS DURIEZ, JOSE EDUARDO WESFREID, PMMH (UMR 7636 CNRS - ESPCI - Univ. Pierre et Marie Curie - Univ. Paris Diderot), GUILLERMO ARTANA, Laboratorio de FluidoDinamica, Universidad de Buenos Aires — The Kelvin-Helmholtz convective instability is the key mechanism in a backward facing step (BFS) flow. The natural flow selects the most amplified mode below the cutoff wavenumber. We introduce time-dependent perturbations inside the shear layer using a DBD plasma actuator. The perturbation is a pulsed low velocity spanwise jet, parallel to the mean flow. It is introduced inside the boundary layer, just upstream separation. Using time-resolved visualizations we show that forcing with a given frequency can have a dramatic effect on the spectral content of the flow and its global properties. The modification of the BFS flow depends strongly on the value of the frequency with respect to the natural frequency. Using 4Hz Particle Image Velocimetry (PIV) we study the evolution of time-averaged properties such as the recirculation length or the shear layer thickness. Recording the forcing perturbation signal together with the PIV acquisition signal we achieve phase-average reconstruction based on the forcing frequency. The phase-averaged time series give access to the wavelength, phase velocity and the spatial distribution of the vortex shedding. It also allows us to build the stability diagram of the shear layer.

10:56AM HF.00003 Sensitivity of an Asymmetric 3D Diffuser to Vortex Generator Induced Inlet Condition Perturbations, EMILY SAYLES, Stanford University, SVEN GRUNDMANN, Center of Smart Interfaces, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Experiments were performed to investigate the flow in an asymmetric 3D diffuser that is highly sensitive to inlet condition perturbations. Previous velocity field measurements showed that in its standard configuration this diffuser develops a stable three-dimensional separation bubble. However, weak secondary flows induced by tunable dielectric barrier discharge plasma actuators in the inlet of the diffuser resulted both in dramatic improvements and degradations in the diffuser’s performance. Two configurations of vortex generators were selected based on their having analogous effects on the diffuser’s pressure recovery. These vortex generators were placed in the inlet of the diffuser and magnetic resonance velocimetry was used to obtain three-dimensional velocity data. The data reveal markedly different separation bubble structures, with the improved pressure recovery corresponding to a reduced reversed flow area. Additionally, the vortex generators which improve the diffuser’s performance create a more uniform velocity profile at the end of the expanding section, while the other configuration facilitates the persistence of a high velocity core through the diffuser’s outlet.
11:09 AM HF.00004 Low-frequency dynamics in supersonic shock/turbulent boundary layer interactions¹, JOHAN LARSSON, JOSEPH W. NICHOLS, Center for Turbulence Research, Stanford University, MATTEO BERNARDINI, SERGIO PIROZZOLI, Dept. of Mech. and Aerosp. Eng., University La Sapienza, Rome, BRANDON E. MORGAN, SANJIVA K. LELE, Dept. of Aero. and Astro., Stanford University — Supersonic shock/turbulent boundary layer interactions are studied using a large LES database. Consistent with previous experimental and numerical findings, the simulations indicate the occurrence of low-frequency dynamics, mainly related to the oscillation of the reflected shock foot. In the proximity of the latter, the wall pressure spectra exhibit substantial spectral content at frequencies at least two orders of magnitude smaller that those typical of the incoming boundary layer. Such trend becomes more pronounced for the strong interactions, with significant flow separation. The analysis of the low- and high-pass filtered flow fields allows to isolate two basic modes: i) a low-frequency ‘breathing’ mode of the separation bubble/ reflected shock system, which is weakly coupled with the upstream flow; and ii) a high-frequency mode, which is dominated by the system response to the incoming boundary layer turbulence. The analysis of the Koopman modes (eigenmodes of the Koopman operator associated with snapshots of the flow field) confirms such scenario, and shows that the wall pressure signature in the low-frequency range is primarily affected by the breathing mode.

¹This work was sponsored by the Center for Turbulence Research, Stanford University through a 2010 Summer Program Fellowship

11:22 AM HF.00005 Global stability analysis of supersonic shock/turbulent boundary layer interactions¹, MATTEO BERNARDINI, SERGIO PIROZZOLI, Dept. of Mech. and Aerosp. Eng., University La Sapienza, Rome, JOSEPH W. NICHOLS, JOHAN LARSSON, Center for Turbulence Research, Stanford University, BRANDON E. MORGAN, SANJIVA K. LELE, Dept. of Aero. and Astro., Stanford University — The global stability of supersonic shock/turbulent boundary layer interactions is investigated in a wide range of shock intensities. The analysis relies on linearization of the governing equations about the mean turbulent flow obtained from LES calculations. The global stability analysis relies on the Arnoldi iterative method, whose convergence requires suitable preliminary smoothing of the base flow. Results of two-dimensional stability analysis highlight the occurrence of a single exponentially growing, zero-frequency mode, also observed by previous investigators. The dominance of such mode in the linearized flow dynamics is confirmed by three-dimensional calculations with slightly disturbed initial conditions. However, the global stability analysis also shows the occurrence of a marginally stable oscillatory mode, whose characteristic frequency is close to those found in LES. Such mode becomes less stable as the incident shock strength increases, and it features a typical ‘breathing’ motion of the recirculation bubble, whereby fluid is periodically entrapped and released, also driving the motion of the reflected shock.

¹This work was sponsored by the Center for Turbulence Research, Stanford University through a 2010 Summer Program Fellowship

11:35 AM HF.00006 DNS of Turbulent Boundary Layer Subject Strong Adverse Pressure Gradient², GUILLERMO ARAYA, Swansea University, LUCIANO CASTILLO, Rensselaer Polytechnic Institute — Direct Numerical Simulations of spatially evolving turbulent boundary layers with prescribed strong adverse pressure gradients are performed. The driven force behind this investigation is to analyze the interaction between the inner and outer layers in adverse pressure gradient with eventual separation. A method for prescribing realistic turbulent velocity inflow boundary conditions is employed. The approach is based on the rescaling-recycling method proposed by Lund et al. (1998) and the dynamic multi-scale method developed recently by Araya et al. (2009). The standard rescaling process requires prior knowledge about how the appropriate velocity and length scales are related between the inlet and recycle stations (e.g. classic scaling laws). Here a dynamic approach is proposed in which such information is deduced dynamically by involving an additional plane located between the inlet and recycle stations. The approach also distinguishes between the inner and outer regions of the boundary layer and enables the use of multiple velocity scales. This flexibility allows applications to boundary layer flows with arbitrary pressure gradients.

²NSF (CBET-0829020) and ONR (R. Joslin)

11:48 AM HF.00007 DNS study of a separation bubble in a turbulent boundary layer, HIROYUKI ABE, YASUHIRO MIZOBUCHI, YUICHI MATSUO, Japan Aerospace Exploration Agency — Direct numerical simulations (DNSs) of a separated turbulent boundary layer have been performed in which blowing and suction are imposed at the upper boundary in order to produce a separation bubble. The inlet data are prescribed by DNSs of a zero-pressure gradient turbulent boundary layer with the rescaling-recycling method. The Reynolds numbers at the inlet are set to be $Re_M = 300$ and 600 where $Re_M$ is the Reynolds number based on the free stream velocity and the momentum thickness. Particular attention is given to the difference between large and small separation bubbles and also the Reynolds-number dependence. The present results indicate that large-scale structures of velocity and pressure fluctuations are more dominant in a large separation bubble than in a small separation bubble, which becomes more apparent with increasing Reynolds number. The relationship between the large-scale structures and the oscillatory behavior of the detachment and reattachment regions will also be discussed.

12:01 PM HF.00008 Study of interacting shear layers in the formation of grid turbulence, JOSE IGNACIO CARDESA-DUENAS, TIMOTHY NICKELS, University of Cambridge — Grid turbulence has been studied for many years as a method of producing approximately homogeneous isotropic turbulence to test classical theories. These studies have concentrated on the region well downstream of the grid since this is where the turbulence is supposed to take its classical form. The way in which the turbulence develops to this state from what is essentially the merged wakes of a series of rods is not yet well understood. This has particular implications as to the extent to which the turbulence downstream is independent of the grid’s geometry. A related question is to what extent inhomogeneity due to the grid persists downstream. In order to study these questions, we fully map the first 15 mesh lengths in the wake of two different biplanar grids using 2D high-speed PIV. This is done with 3 cameras to cover a large field of view in the downstream direction with adequate resolution. Measurements are taken for each grid at three different Reynolds numbers ($Re_M$ up to 1600) and on two different planes with respect to the grid: half way between two rods and exactly behind a rod.

12:14 PM HF.00009 PIV Measurements of the Near-Wake behind a Fractal Tree², KUNLUN BAI, CHARLES MENEVEAU, JOSEPH KATZ, Johns Hopkins University — An experimental study of turbulent flow in the wake of a fractal-like tree has been carried out. Fractals provide the opportunity to study the interactions of flow with complicated, multiple-scale objects, yet whose geometric construction rules are simple. We consider a pre-fractal tree with five generations, with three branches and scale-reduction factor 1/2 at each generation. Its similarity fractal dimension is $D_s \sim 1.585$. Experiments are carried out in a water tunnel with the ability of index-matching, although current measurements do not utilize this capability yet. The incoming velocity profile is designed to mimic the velocity profile in a forest canopy. PIV measurements are carried out on 14 horizontal planes parallel to the bottom surface. Drag forces are measured using a load cell. Mean velocity and turbulence quantities are reported at various heights in the wake. Mean vorticity contours on the upper planes show signatures of the smaller branches, although the wakes from the smallest two branches are not visible in the data possibly due to rapid mixing. Interestingly, their signatures can be observed from the elevated spectra at small scales. Momentum deficit in the wake profiles and drag forces are compared. The results from this experiment also serve as database against which to compare computer simulations and models.

²Supported by NSF - ATM 0621396.
12:27PM HF.00010 PIV Analysis Comparing Aerodynamic Downforce Devices on Race Car in Water Tunnel , SAM HELLMAN, PETER TKACIK, MESBAH UDDIN, SCOTT KELLY, University of North Carolina at Charlotte — There have been claims that the rear wing on the NASCAR Car of Tomorrow (COT) race car causes lift in the condition where the car spins during a crash and is traveling backwards down the track at a high rate of speed. When enough lift is generated, the race car can lose control and even fly off of the track surface completely. To address this concern, a new rear spoiler was designed by NASCAR to replace the wing and prevent this dangerous condition. Flow characteristics of both the rear wing and the new spoiler are qualitatively analyzed using particle image velocimetry (PIV). The experiment is done in a continuous flow water tunnel using a simplified 10% scale model COT. Flow structures are identified and compared for both the wing and spoiler. The same conditions are also reviewed when the car is traveling backwards as it might during a crash. The cause of the lift generated by the rear wing when in reverse is shown.

Monday, November 22, 2010 10:30AM - 12:40PM – Session HG GFD: Atmospheric Flows II Long Beach Convention Center 103B

10:30AM HG.00001 Microscale Measurement in the Atmospheric Boundary Layer: Collapse of Turbulence1 , HARRINDRA FERNANDO, University of Notre Dame, ELIEZER KIT, Tel Aviv University, ANN DALLMAN, University of Notre Dame — During cooling of the Earth’s surface in the evening, day-time convection subsides due to cut-off of its energy sources and a stable density stratified layer develops near the ground. In complex terrain, this evening transition from the convective boundary layer to the stable boundary layer is associated with low wind speeds, and hence low shear production of turbulence. Often the wind direction is also variable during the evening transition, and hence the use of probes such as hotwires/films for the measurements of microscale turbulent quantities needs special handling as they require the winds to have a specific alignment with the probe. To circumvent this problem, a combo of co-located sonic and hot-film anemometers, with the former measuring mean winds and aligning the latter in the appropriate wind direction via an automated platform, was successfully designed and implemented. A novel calibration procedure for the probes was also developed. It was found that the evening transition in complex terrain is associated with a sudden collapse of turbulence spectra across the entire spectrum. Observations taken in multiple locations show that the collapse is a complex phenomenon, sometimes showing layering with low rms vertical velocities and in other times showing higher vertical velocities perhaps due to instabilities and billowing.

1Supported by NSF/US-Israeli Bi-National Science Foundation

10:43AM HG.00002 Jupiter’s Zonal Winds: Are They Bands of Homogenized Potential Vorticity and Do They Form a Monotonic Staircase? , PHILLIP MARCUS, University of CA at Berkeley, SUSHIL SHETTY, Schlumberger-Doll Research — It has been hypothesized that the potential vorticity (PV) in Jupiter’s atmosphere is mixed in a manner that is analogous to the Phillips effect in the ocean. When the upper ocean is mixed, the salt density distribution changes from a smoothly increasing function of depth to a nearly monotonic staircase with regions of nearly uniform salt density separated from each other by sharp interfaces where the density gradient is large. It is hypothesized that the profile of PV in Jupiter’s east-west zonal winds (visible stripes) is a staircase, decreasing from north to south. Measurements of the Jovian zonal velocity are sufficiently precise to determine vorticity, but the PV also depends on unknown parameters that cannot be observed directly. Therefore, the distribution of PV cannot be tested directly. By using new high-precision observations of Jupiter, we have solved numerically the inverse problem between the latitudes of 19°S and 39°S and found the PV (and its uncertainties) that best fits the observations. Although we find that the PV distribution is approximately piecewise-constant, the zonal PV is monotonic. We show that this non-monotonicity is necessary to make the Great Red Spot nearly round (aspect ratio of 1.6), and that without the non-monotonicity, the Red Spot would be highly elongated in the east-west direction and probably unstable.

10:56AM HG.00003 ABSTRACT WITHDRAWN –

11:09AM HG.00004 Lagrangian coherent structures and transport in hurricanes , DOUG LIPINSKI, KAMRAN MOHSENI, University of Colorado - Boulder — Hurricane intensity forecasting remains one of the most difficult challenges in weather research. At present, there is a lack of understanding with regards to the appropriate oceanic boundary conditions for the hurricane and the corresponding energy and moisture transport. In this talk, the Lagrangian coherent structures (LCS) present in a numerical simulation of hurricane Rita (2005) are identified. The LCS reveal the low level atmospheric transport in this hurricane and provide insight into the conditions which may strengthen or weaken the hurricane. This information may be used to better focus future research efforts in this area by illuminating the key mechanisms for transport in the low level atmosphere of a hurricane.

11:22AM HG.00005 Stable stratification in turbulent Ekman layers , OSCAR FLORES, JAMES RILEY, University of Washington, NICHOLAS MALAYA, ROBERT MOSER, University of Texas at Austin — In order to study the day to night transition in the atmospheric boundary layer we perform DNS of turbulent Ekman layers whose height (h) is prescribed by an overlying inversion. We will present results from several simulations, with friction Reynolds numbers up to Re = u,w h/ν = 2800 depending on the domain size. In all cases, a quasi-steady state is reached with an adiabatic boundary condition at the ground. Then a constant negative heat flux is applied at the ground, to mimic the radiative cooling of the ground during clear sky nights. The results indicate that the buffer region locally collapses when Lr(3/2) < 100, where L is the Monin–Obukhov lengthscale. In the outer region of the flow, eddies with sizes larger than L are damped by the stratification in times of the order of their eddy turn-over time, even if at those times the mean temperature profile is relatively shallow. These results are consistent with Monin’s self-similar theory and with both experimental and field observations. Funded by ARO Grant No. W911NF-06-1-0155 and NSF Grant No. OCI- 0749209.

11:35AM HG.00006 Stability of a stratified boundary layer flow when shear and stratification are not aligned , JULIEN CANDELIER, CEA, DAM, CIF, STEPHANE LE DIZES, IRPHE, CNRS, Aix-Marseille University, CHRISTOPHE MILLET, CEA, DAM, CIF — The inviscid stability properties of a boundary layer flow with a tanh velocity profile in a stably stratified fluid with a constant Brunt-Väisälä frequency is examined when the direction of the shear is inclined with an angle  with respect to the vertical direction of stratification. We show that for all Froude numbers there exists a critical angle  above which the boundary layer becomes inviscidly unstable. The characteristics of the unstable modes are analysed. For small Froude numbers, unstable modes are shown to be 3D radiative modes with a inner wave structure that extends to infinity. For large Froude numbers, the modes are localized in the boundary layer and their frequency and growth rate are proportional to the Brunt-Väisälä frequency. Non-Boussinesq compressibility effects on the stability properties are also considered. The results are discussed in the context of atmospheric applications.

11:48AM HG.00007 Internal waves crossing an interface , JOHN MCHUGH, University of New Hampshire — Internal waves in continuously stratified flow in two layers is considered. The interface between layers is defined by a jump in the Brunt-Väisälä frequency, assumed constant in each layer. The density profile is chosen to be continuous across the interface, and the flow is assumed to be Boussinesq. The waves are periodic in the horizontal but modulated in the vertical. A weakly nonlinear approach produces three amplitude equations with cubic nonlinearity, one for incident, reflected, and transmitted wave packets. The results show that a wave-induced mean flow is strongest near the interface and underneath it. Furthermore, this mean flow is discontinuous, and has an oscillatory component. These results provide a likely scenario for higher levels of atmospheric turbulence near Earth’s tropopause and other similar interfaces.
12:01PM HG.00008 Synchronization of modulated travelling baroclinic waves in thermally driven rotating annulus flows

ALFONSO A. CASTREJON PITÁ, PETER L. READ. Clarendon Laboratory, Department of Physics, University of Oxford, Parks Road, OX1 3PU, Oxford, UK — Synchronization in a fluid dynamical analogue of atmospheric circulation is studied experimentally by investigating the dynamics of a pair of rotating annulus systems, coupled in real time via their thermal boundary conditions, in both periodic and chaotic regimes. The combined effects of differential heating in the horizontal and background rotation leads to the formation of a zonally-symmetric baroclinic jet flow that may become unstable under some conditions to travelling baroclinic waves which may be steady or modulated in amplitude, and a range of more complex spatiotemporal flows. Synchronization tools such as phase analysis and frequency locking are used to study the resulting dynamics of the coupled system and, depending upon the coupling configuration (master-slave or bidirectional), coupling strength and parameter mismatch, demonstrate various degrees of synchronization including partial/imperfect phase and complete phase synchronization (at various frequency ratios). These results suggest the possible importance of synchronization in natural climate variability since the studied coupled system forms a direct analogue of coupled weather systems in different locations in the atmosphere on seasonal and intraseasonal timescales.

12:14PM HG.00009 Radiative instability in stratified rotating flows1. STEPHANE LE DIZES, XAVIER RIEDINGER, PATRICE MEUNIER, IRPHE, CNRS, Aix-Marseille University.— We present new theoretical and experimental works which demonstrate the existence of an instability associated with the emission of internal waves in rotating flows when they are stably stratified along their rotation axis. A comprehensive stability diagram is obtained for both a potential flow and a Keplerian flow defined by their angular velocity $\Omega_r(x) = 1/r^2$ and $\Omega_k(r) = 1/r^{3/2}$ respectively in an infinite domain ($r \in (1, \infty)$) as functions of the Rossby number (background rotation) and the Froude number (strength of the stratification along the rotation axis). Both flows are shown to become unstable in centrifugally stable regimes thanks to the stratification. Experimental evidence is also provided for the potential flow around a rotating cylinder. Measurements of the characteristic wavelength and frequency are compared to the theory and a good agreement is demonstrated. The same instability is shown to be active in compressible or shallow-water flows where acoustic waves or surface waves can be emitted.

1The French National Agency for Research (grant # BLAN06-3-137005) is acknowledged.

12:27PM HG.00010 Temporal behavior and vortex topology of topographic wave-breaking. OLIVIER EIFF, U. Toulouse, CNRS-IMFT, NICOLAS BOULANGER, KARINE LEROUX, ALEXANDRE PACI, Meteo-France/CNRM, CNRS-GAME — At low Froude numbers or strong stratification, the internal waves generated by flow over an obstacle or mountain will overturn and break. Surprisingly little is known, however, of the dynamics of the wave breaking itself. Anasyev and Pelletier (1998) investigated the wave breaking region via LES and Eiff et al. (2005) via PIV measurements, but both presumed a statistically stationary wave-breaking process after the initial wave overturning. Here, we propose to take a closer look at this assumption by closely analyzing the spatio-temporal structure of internal wave breaking region and the surrounding flow. The analysis is based on Hovmöller diagrams and spatial correlations obtained from 2D-PIV measurements of flows generated in uniform stratified flow over 2D and quasi-2D obstacles in salt-stratified hydrodynamic channels at different Reynolds numbers ranging from laminar to turbulent. The results reveal low frequency variations throughout the flow field, in and outside the wave-breaking region. This characteristic frequency can be related to be due to a sequence of growth and decay of wave-breaking. Finally, new 3D-3C PIV measurements at high Reynolds numbers reveal a first glimpse of the 3D vortex topology.

Monday, November 22, 2010 10:30AM - 12:40PM - Session HH Convection and Buoyancy Driven Flows V Long Beach Convention Center 103C

10:30AM HH.00001 Lagrangian particle tracking in turbulent convection. VALENTINA LAVEZZO, FEDERICO TOSCHI, HERMAN J.H. CLERCX, Eindhoven University of Technology — The dispersion of inertial particles in turbulent convection has direct relevance for many industrial and environmental applications, where the fluid heat transfer can be modified by the presence and the deposition of particles at the walls (e.g. nuclear power plants, petrochemical multiphase reactors, cooling systems for electronic devices, pollutant dispersion in the atmospheric boundary layer, aerosol deposition etc.). A high resolution numerical technique coupled with Lagrangian particle tracking is employed, in this work, to investigate the behaviour of inertial particles in a periodic turbulent Rayleigh-Bénard convection cell. In particular, we focus on the effects of different flow regimes, obtained varying the Rayleigh number, on particle dispersion/resuspension. Different Stokes numbers are considered to evaluate the influence of inertia on particle clustering and consequently, on the heat exchange modification between the two walls. Single and two particle statistics are used to estimate the level of mixing and the role of turbulent structures in particle transport. Mean and higher order statistics on particle and fluid velocity and temperature fields are also presented.

10:43AM HH.00002 An Experimental Study of Lagrangian Statistics in Rayleigh-Bénard Convection1. RUI NI, SHI-DI HUANG, SHENG-QI ZHOU, KE-QING XIA, The Chinese University of Hong Kong — We present an experimental study of Lagrangian statistics in Rayleigh-Bénard convection, using water as the working fluid. The tracking volume is (5 cm)3 in the centre of a cylindrical shaped convection cell of aspect ratio one and 20 cm in height. Three cameras were used to identify the 3-dimensional positions of the tracer particles, which were evenly suspended in the cell. Detailed properties of the particle velocity, acceleration and dispersions along different directions have been investigated. We also studied the dependency of pair properties on the initial pair separation. All these properties have been examined with Rayleigh number spanning from 108 to 1010 and Prandtl number around 6.2.

1Work supported by the Research Grants Council of Hong Kong SAR (Project Nos. CUHK 403806).

10:56AM HH.00003 An alternative Lagrangian approach to laminar heat transfer. MICHEL SPEET-JENS, Eindhoven University of Technology — Heat transfer in essence is the transport of thermal energy along certain paths in a similar way as fluid motion is the transport of fluid parcels along fluid paths. This similarity in principle admits Lagrangian heat-transfer analyses in terms of the geometry of such “thermal paths” analogous to the well-known Lagrangian analyses on chaotic mixing in viscous flows and micro-fluidics. To date such Lagrangian approaches towards laminar heat transfer represent convective heat transfer by the enthalpy flux. However, though conceptually entirely correct, this ansatz hampers physical interpretation of Lagrangian heat-transfer analyses, as the enthalpy is determined only up to a uniform background state. An alternative approach is proposed that may resolve this indeterminacy. This approach is outlined and demonstrated for the laminar heat transfer in a simple 2D unsteady flow.

11:09AM HH.00004 A variational principle for solute fluxes in mushy-layer convection. ANDREW WELLS, JOHN WETTLaUFER, STEVEN ORSZAG, Yale University — The utility of variational principles to describe nonlinear dissipative systems has been a topic of long-standing debate. We apply a variational principle to describe nonlinear convection in a mushy layer: a reactive porous medium formed during solidification of a binary alloy. Convection drives the formation of channels of zero solid fraction, or chimneys, which are the principle conduits through which solute drains from the mushy layer. By optimizing the rate of removal of stored potential energy, our numerical model predicts scalings for solute fluxes and chimney spacings consistent with previous simulations and laboratory experiments. This leads to predictions of solute fluxes from growing sea ice.
11:22AM HH.00005 The Steepling of Mushy Layers1. ANTHONY ANDERSON, GRAE WORSTER, DAMTP, University of Cambridge — The rapid solidification of a binary alloy leads to the formation of a mushy layer, comprised of a dendritic solid phase and a concentrated interstitial fluid phase. When freezing from below, such that the mean density field is statically stable, a phenomenon known as “steepling” has been observed, whereby the mushy layer becomes domed. In experiments, the degree of steepling has been shown to increase with a decreased rate of solidification and the lateral extent of the steeple is comparable to the size of the container. It was reasoned that steepling is the cause of an instability of a planar front induced by convection within a proposed compositional boundary layer at the base of the mushy layer in an otherwise stable solute field. We explore this possibility using a linear stability analysis of the mush-liquid interface and compare the results to unidirectional solidification experiments using aqueous NaCl solutions.

1Based upon work supported by the National Science Foundation under Award No. OISE-0965138.

11:35AM HH.00006 Natural convection in a square cavity with participating medium . MANUEL ALEJANDRO RAMÍREZ CABRERA, EDUARDO RAMOS, CIE-UNAM — The natural convective flow in a two dimensional square cavity filled with a material which has properties of an optical participating medium is theoretically analyzed. Radiant energy coming from an external heat source is assumed to fall on a small region of one of the lateral walls of the cavity, and as the working fluid is assumed to be participating, the incoming energy is absorbed in its volume, heating the material by conduction, convection and radiation. The simultaneous presence of temperature gradients and a body force generates a convective motion. We present a mathematical model for describing this phenomenon which includes the conservation equations of mass, momentum and energy. The integral term that describes the radiation heat transport is included in the energy conservation equation. The solution is obtained with a numerical method and representative cases are described. This study has potential applications in the design of heat exchangers in central solar towers.

11:48AM HH.00007 Time-periodic traveling solutions of localized convection cells and their collision in binary fluid mixture1. TAKEI WATANABE, Research Institute for Electronic Science, Hokkaido University, KAZUTAKA TOYABE, Hokkaido University Graduate School of Science, MAKOTO IMA, YASUMASA NISHIYI, Research Institute for Electronic Science, Hokkaido University — We study the mathematical structure of localized convection cells in a binary fluid mixture, some of which are not observed in Rayleigh-Bénard convection in a pure fluid. In particular, a solution representing time-periodic traveling localized convection cells (periodic traveling pulse, PTP) has not been obtained even numerically because this solution requires two unknown variables to be determined: group velocity and temporal period in the convective frame with the group velocity. We developed a new integrated numerical method to obtain the PTP solution as well as the steady, periodic, and traveling solutions. By using this method, a global bifurcation structure containing a variety of solutions including PTPs is obtained and the phase dependence of the collision of counter-propagating PTPs is investigated in detail.

1A part of this study is supported by Grant-in-Aid for Scientific Research (KAKENHI) No. 21340019 and Core Research for Evolutional Science and Technology (CREST) No. PJ74100011.

12:01PM HH.00008 Convective instabilities in a ferrofluid with a viscoelastic carrier fluid. HARALD PLEINER, Max Planck Institute for Polymer Research, Mainz, Germany, JAVIER MARTÍNEZ-MARDONES, Instituto de Física, Pontificia Universidad Católica de Valparaíso, Chile, LAURA PEREZ, Departamento de Ingeniería Metalúrgica, Universidad de Santiago de Chile, DAVID LAROZE, Max Planck Institute for Polymer Research, Mainz, Germany, Instituto de Alta Investigación, Universidad de Tarapacá, Arica, Chile — We report theoretical and numerical results on the convective instability for a ferrofluid in a viscoelastic carrier fluid. Such a system exhibits several features that are important for the onset and development of convective instabilities, like the Soret or thermofluidic effect due to the binary mixture nature, shear thinning or thickening, stress retardation and normal stress generation due to the viscoelasticity of the carrier liquid, and the Kelvin force and magnetophoresis due to the ferromagnetic structure of the colloidal particles. Convective instabilities can be triggered by applying temperature (concentration) gradients and/or (homogeneous) magnetic fields. We systematically investigate the role of the various effects (and their mutual interplay) for the instability and bifurcation behavior. In particular, for oscillatory instabilities nonlinear magnetic and nonlinear viscoelastic properties are taken into account.

12:14PM HH.00009 Dynamical behavior of lean swirling premixed flame generated by change in gravitational orientation. HIROSHI GOTODA, TAKAYA MIYANO, Ritsumeikan University, IAN SHEPHERD, Lawrence Berkeley National Laboratory — The dynamic behavior of flame front instability in lean swirling premixed flame generated by the effect of gravitational orientation has been experimentally investigated in this work. When the gravitational direction is changed relative to the flame front, i.e., in inverted gravity, an unstably fluctuating flame (unstable flame) is formed in a limited domain of equivalence ratio and swirl number (Gotoda. H et al., Physical Review E, vol. 81, 026211, 2010). The time history of flame front fluctuations show that in the buoyancy-dominated region, chaotic irregular fluctuation with low frequencies is superimposed on the dominant periodic oscillation of the unstable flame. This periodic oscillation is produced by unstable large-scale vortex motion in combustion products generated by a change in the buoyancy/swirl interaction due to the inversion of gravitational orientation. As a result, the dynamical behavior of the unstable flame becomes low-dimensional deterministic chaos. Its dynamics maintains low-dimensional deterministic chaos even in the momentum-dominated region, in which vortex breakdown in the combustion products clearly occurs. These results were clearly demonstrated by the use of nonlinear time series analysis based on chaos theory, which has not been widely applied to the investigation of combustion phenomena.

12:27PM HH.00010 Three-dimensional simulations of burning thermahs. ANDY ASPDEN, JOHN BELL, Lawrence Berkeley National Lab, STAN WOOSLEY, UC Santa Cruz — Flame ignition in type Ia supernovae (SNe Ia) leads to isolated bubbles of burning buoyant fluid. As a bubble rises due to gravity, it becomes deformed by shear instabilities and transitions to a turbulent buoyant vortex ring. Morton, Taylor and Turner (1956) introduced the entrainment assumption, which can be applied to inert thermals. In this study, we use the entrainment assumption, suitably modified to account for burning, to predict the late-time asymptotic behaviour of these turbulent buoyant vortex rings in SNe Ia. The theory is validated against three-dimensional simulations with adaptive mesh refinement at effective resolutions up to 4096.

Monday, November 22, 2010 10:30AM - 12:27PM Session HJ Flow Control V Long Beach Convention Center 201A

10:30AM HJ.00001 High-Speed Jet Noise Source Identification by Wavelet Filtering. JACQUES LEWALLE, ZACHARY BERGER, KERWÍN LOW, MARK GLAUSER, Syracuse University — Pressure sensors in a Mach 0.6 jet provide near-field data at 2 sections (x/D = 3 and 6), and simultaneous far-field data at 5 angular locations from 15° to 90° degrees relative to the jet axis. Continuous wavelets allow some feature recognition at the various stations. In the absence of sustained oscillations, the Mexican hat wavelet is used on the Fourier mode 0 in the near-field. The physical relevance of this decomposition is established by the cross-correlation of the filtered near-field data with far-field noise. A scale-dependent cross-correlation was calculated, showing distinct scales and convection propagation delays for the various pairs of traces, for which different causes will be discussed. At the time of writing, the distinctive characteristics are used for pattern recognition in the raw data.
10:43AM HJ.00002 Shear Layer Excitation of a High Speed Turbulent Jet, KERWIN LOW, BASMAN EL HADIDI, MARK GLAUSER, ZACHARY BERGER, Syracuse University — Simultaneous pressure and acoustic measurements are acquired in the hydrodynamic and acoustic fields of a Mach 0.6 cold jet (Re = 680,000). The two axisymmetric sensor arrays in the near-field (x/D = 3 and 6) are positioned 10cm from the developing shear layer. The far-field microphones (x/D = 75) are located at five angular locations from 15° to 90° relative to the jet axis. Presently, these experiments are set up to characterize the system response of the near field jet shear layer to different modes of forcing. Several open and closed loop control tests were conducted. The open loop control cases included simple sinusoidal forcing (with varying coefficient of moment and frequency), phased forcing and amplitude modulated forcing. The closed loop cases included feedback from the Fourier filtered signals from 3 diameters and 6 diameters downstream. The correlations between the near field Fourier filtered pressure modes and the far field noise are changed significantly for all control cases; demonstrating the ability to effect control authority in the near field region. An examination of the far field noise spectra, however, demonstrates only minor changes from the control.

10:56AM HJ.00003 Three Dimensional Vortex Formation of a Finite-Span Synthetic Jet, TYLER VAN BUREN, MICHAEL AMITAY, Rensselaer Polytechnic Institute. Synthetic jets have been a topic for multiple investigations in the field of flow control. Understanding the flow physics associated with these jets is crucial in the development and application of this technology. Synthetic jets are commonly used in boundary layer control, and a large part of that comes from the flow interaction with the vortices created at the synthetic jet orifice. Three dimensional particle image velocimetry experiments have been conducted on the flow field near the orifice of a synthetic jet issued into a quiescent flow with interests in exploring the effects of multiple geometric features (such as throat length, aspect ratio, exit angle, etc.) of a rectangular orifice. Supported by Boeing

11:09AM HJ.00004 Eliminating turbulence in spatially intermittent flows, BJOERN HOF, ALBERTO DE LOZAR, MARC AVILA, Max-Planck-Institute for Dynamics and Self-Organization, TOBIAS SCHNEIDER, Harvard University — When transferring large quantities of fluid it is energetically far more efficient if the fluid motion is laminar since here friction losses are low. Flows through pipes and channels are sensitive to minute disturbances even at moderate velocities and in practice most flows are turbulent. We here isolate an amplification mechanism which constantly feeds energy from the mean shear into turbulent eddies. In pipe and channel experiments a simple control strategy is applied to interrupt this energy transport at intermediate flow-rates. When activated an immediate collapse of turbulence is observed and the flow re-laminarises. While in experiments this simple method is limited to moderate Reynolds numbers, numerical simulations show that the same principle works at much larger Re. Possible extensions to experiments at higher flow rates are discussed.

11:22AM HJ.00005 Separation control in a conical diffuser with an annular inlet, KIN PONG LO, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Conical diffusers are commonly used in turbomachines to slow down the flow and recover pressure. In typical applications such as the diffusers in a power turbine, the inlet to the diffuser is annular. A large central separation bubble forms if the central hub ends abruptly. A long streamlined tail cone can eliminate the separation, but it is often unfeasible for structural reasons. Experiments were performed to investigate various means to manage both the central separation bubble and any separation on the outer diffuser walls. The Reynolds number is 66000 based on the annulus bulk velocity and hydraulic diameter. Full-field, three-component velocity data were measured using magnetic resonance velocimetry. The central separation bubble behind the hub extends the full length of the diffuser in the absence of any control. A Coanda jet at the end of the hub can strongly reduce or completely eliminate the central separation bubble. However this can cause separation from the conical diffuser walls in some cases. A step in the outer diffuser wall acts to fix the location of separation making it more amenable to control. Several control mechanisms for this outer separation bubble are under investigation.

1Siemens Power Generation

11:35AM HJ.00006 Experimental Investigation of Actuators for Flow Control in Inlet Ducts, JOHN VACCARO, YOUSEF ELIMELECH, MICHAEL AMITAY, Rensselaer Polytechnic Institute, FLOW CONTROL RESEARCH LAB TEAM — Attractive to aircraft designers are compact inlets, which implement curved flow paths to the compressor face. These curved flow paths could be employed for multiple reasons. One of which is to connect the engine intake to the engine embedded in the aircraft body. A compromise must be made between the compactness of the inlet and its aerodynamic performance. The aerodynamic purpose of inlets is to decelerate the oncoming flow before reaching the engine while minimizing total pressure loss, unsteadiness and distortion. Long length-to-diameter ratio inlets have a high degree of curvature, which inevitably causes flow separation and secondary flows. Currently, the length of the propulsion system is constraining the overall size of Unmanned Air Vehicles (UAVs), thus, smaller more efficient aircrafts could be realized if the propulsion system could be shortened. Therefore, active flow control is studied in a compact (L/D=1.5) inlet to improve performance metrics. Actuation from a spanwise varying coanda type ejector actuator and a hybrid coanda type ejector / vortex generator jet actuator is investigated. Special attention will be given to the pressure recovery at the AIP along with unsteady pressure signatures along the inlet surface and at the AIP.

1Support from Northrop Grumman Corporation

11:48AM HJ.00007 On the lower bound of net driving power in controlled duct flows, KOJI FUKAGATA, Keio University, KAZUYASU SUGIYAMA, NOBUHIDE KASAGI, The University of Tokyo — We examine mathematically the lower bound of the net driving power of a controlled flow under a constant flow rate. The net power in a duct with arbitrary cross-section in the presence of the inertial term, blowing/suction from the wall, and arbitrary body forces can be decomposed into four terms: (1) dissipation due to the velocity profile of Stokes flow; (2) dissipation due to deviation of mean velocity from the Stokes flow profile; (3) dissipation due to velocity fluctuations; and (4) correlation between the wall-pressure of Stokes flow and the time-averaged blowing/suction velocity. Among these, the first three terms are shown to be non-negative, while the sign of the fourth term is indefinite. The fourth term vanishes in the cases where the duct has a constant-shape cross-section, such as circular pipes and plane channels. Namely, in such cases, the lower bound of net power is exactly given by the dissipation rate of the Stokes flow at the same flow rate.

12:01PM HJ.00008 Optimal localized control of the onset of turbulence in a channel flow, RASHAD MOARREF, BINH K. LIEU, MIHAIO R. JOVANOVIC, University of Minnesota — For the problem of controlling the onset of turbulence in a channel flow, we study the design of optimal localized state-feedback controllers. The actuation is generated by blowing and suction at the walls and we assume that (i) the actuators are placed along a two-dimensional lattice of equally spaced points; and that (ii) each actuator uses information from only a limited number of nearby sensors. We utilize recently developed tools for designing structured optimal feedback gains to reduce receptivity of velocity fluctuations to flow disturbances in the presence of control. Our preliminary DNS results, conducted at low Reynolds numbers, show that this approach can indeed maintain the laminar flow. This is in contrast to the localized strategies obtained by spatial truncation of optimal centralized controllers, which may introduce instability and promote transition even in the situations when the uncontrolled flow stays laminar.

1Part of this work was performed during the 2010 Summer Program at the Center for Turbulence Research with financial support from Stanford University and NASA Ames Research Center.
In contrast to previous studies, we find the evolution of the ejecta radius approaches a $t^{1/2}$ scaling law, as seen for the crown evolution in liquid drop impacts. The velocity of small particles. The velocity of the first ejected grains increase with the kinetic energy of the impacting sphere. We also observe that the fast grains, which can obtain velocities up to 8 times that of the impacting sphere, emerge at the lowest ejection angles. As the grain size is decreased, the velocities of the first ejected grains increase with the kinetic energy of the impacting sphere. We also observe that the direction of shear-driven segregation depends on the nature of the flow itself, collisional or frictional.

10:56AM HK.00003 Local Dynamics of Granular Size Segregation, ADAM KEITH, JAMES PUCKETT, KAREN DANIELS, NC State University — We seek to quantify the local mechanisms which drive granular size segregation, using a two-dimensional system. We perform experiments using a bi-disperse mixture of disks floating on a tilted plate, agitated by bumps at the bottom edge. A layer of large particles is initially placed at the bottom of the system mix with a layer of small particles above it, eventually segregating to the upper surface. We record the position of each particle and measure the average segregation velocity as a function of local packing fraction $\phi$ for all particle and local concentration $c$ of small particles. The velocity of the large particles is strongly dependent on packing fraction; particles in regions of lower $\phi$ tend to move downward, while those in regions of higher $\phi$ ascend through the material. In contrast, we find that the effect of local concentration $c$ is weak.

11:09AM HK.00004 Density Fluctuations in Vibrating Granular Monolayers, GUSTAVO CASTILLO, NICOLAS MUIJCA, PABLO GUTIERREZ, LORETO OYARTE, Universidad de Chile, SCOTT WAITUKAITIS, University of Chicago, RODRIGO SOTO, Universidad de Chile — This study aims to quantify density fluctuations in a fluidized quasi-two-dimensional granular system close to a solid-liquid-like transition. This transition is reached above an acceleration threshold and at sufficiently high density. The system is a shallow square cell built with two square ITO coated glass plates. The cell is filled with approximately 10000 spherical 1 mm stainless steel particles and the filling density is about 85%. Due to the dissipative nature of grain contacts, energy is injected in the system by vertical vibrations. To characterize the system we measure its Static Structure Factor as well as dynamical correlation functions.

11:22AM HK.00005 Liquid effect on the vibration of granular media in cylindrical cavities, ENRIQUE GUZMAN, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — The study of the interactions of granular media with liquid phases is important both, from the academic applied points of view. A particularly interesting problem concerns the dispersion of the granular phase into the liquid phase. To this end, a series of experiments are being conducted in order to determine the conditions under which such dispersion takes place. The experimental apparatus consists of a short transparent cylinder (LVd) with its axis oriented in a horizontal position. The cavity is completely filled with liquid and a prescribed number of glass spheres forms a deposit layer at the bottom. The cylinder, which is initially at rest, is set into a vertical vibrating state of motion by means of an external actuator. While the amplitude of the excitation remains fixed, its frequency is swept (continuously) from 5Hz to 15Hz. Synchronized high speed imaging is then used to identify the frequency at which the stratified-to-dispersed transition occurs. Preliminary results clearly indicate the essential role played by the properties of the liquid (i.e. density, viscosity and superficial tension) and of the spheres (i.e. size and number) during the process. The objective of the study is to determine the conditions required to produce appropriate dispersions for different combinations of liquids and spheres.

11:35AM HK.00006 High-speed x-ray tomographic imaging of a ball impacting on sand, TESS A.M. HOMAN, Physics of Fluids, University of Twente, Enschede, The Netherlands, EVERT C. WAGNER, ROB F. MUDDE, Multi-Scale Physics, Delft University of Technology, The Netherlands, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids, University of Twente, Enschede, The Netherlands — When a ball is dropped in fine, very loose sand, a cavity is formed inside the sand bed which collapses, creating a jet and entraining an air bubble. At a fixed depth below the surface, the shape and dynamics of a horizontal cross section of the cavity are studied by means of high-speed x-ray tomography system. Repeating the procedure at different depths provides a full time-resolved reconstruction of the cavity within the sand bed. Using this reconstruction we test several hypotheses on the process of sand jet formation.

11:48AM HK.00007 Measuring the effect of air during granular impact events, DEVARAJ VAN DER MEER, TESS HOMAN, Physics of Fluids, University of Twente, The Netherlands, SYLVAIN JOUBAUD, Laboratoire de Physique, CNRS-Université de Lyon, France, DETLEF LOHSE, Physics of Fluids, University of Twente, The Netherlands — Air is known to play a crucial role during the impact of a sphere into a bed of fine, loose sand. This can be traced back to a significant increase of the drag the object experiences inside the sand at low ambient pressures, but what remains unclear is the mechanism by which the drag increases. To shed light upon this mechanism we record the pressure changes during impact, both above and below the bed. From this, with the help of Darcy’s law, we deduce the magnitude of the air flows inside the sand which are caused by the impacting sphere and relate these to the observed drag increase.

12:01PM HK.00008 Emergence of fluid-like granular ejectas generated by sphere impact, JEREMY MARSTON, SIGURDUR THORODDSSEN, KAUST — Experimental data is presented for the speed and shape of the ejecta when a solid sphere impacts onto a granular bed. We use high-speed imaging to provide direct measurement of individual grain velocities and trajectories as well as the overall evolution of the granular ejecta. For larger bed grain sizes, the velocities of the first ejected grains increase with the kinetic energy of the impacting sphere. We also observe that the fastest grains, which can attain velocities up to 8 times that of the impacting sphere, emerge at the lowest ejection angles. As the grain size is decreased, a more ‘fluid-like’ behavior is observed whereby the ejected material first emerges as a thin sheet between the sphere and the bed surface. In this instance, in contrast to previous studies, we find the evolution of the ejecta radius approaches a $t^{1/2}$ scaling law, as seen for the crown evolution in liquid drop impacts.
12:14PM HK.00009 Cooperative penetration in a light granular medium, J. CARLOS RUIZ-SUAREZ, Cinvestav-IPN, Unidad Monterrey, FELIPE PACHECO-VAZQUEZ, Cinvestav-IPN, Unidad Merida — A projectile impacting against a granular medium exemplifies the interesting nature of granular matter. Whether the projectile is an asteroid striking the crust of a planet or an object thrown against a granular bed in the laboratory, once the intruder makes contact with the medium it inevitably encounters a stopping force. The character of this force underscores several fundamental issues, from geological and biological sciences, to soil research and technological applications. The impact velocity dependence, the final penetration depth as a function of different parameters, the nature of the drag force, are nowadays well understood thanks to the recent work carried out by different groups. Furthermore, the effects of confinement, object symmetry and fragility of the medium have also been considered. However, despite all this effort, we know very little about what occurs when more than one intruder impact simultaneously a granular medium. Here we show and discuss some experimental findings about the penetration dynamics followed by a group of intruders impacting a granular medium. The particles used in our study are much lighter than water, therefore, intruders penetrate deeply into the system depicting intriguing cooperative behaviours that hint to hydrodynamic-like interactions.

12:27PM HK.00010 Electrostatics in sandstorms and earthquakes, TROY SHINBROT, NIRMAL THYAGU, Rutgers University, THOMAS PAEHTZ, HANS HERRMANN, ETH, Zurich — We present new data demonstrating (1) that electrostatic charging in sandstorms is a necessary outcome in a class of rapid collisional flows, and (2) that electrostatic precursors to slip events - long reported in earthquakes – can be reproduced in the laboratory.

Monday, November 22, 2010 10:30AM - 12:27PM – Session HL Biofluids: Physiological Circulatory II Long Beach Convention Center 202A

10:30AM HL.00001 Subharmonic response from ultrasound contrast microbubbles for non-invasive blood pressure estimation, AMIT KATIYAR, KAUSIK SARKAR, University of Delaware, FLEMMING FORSBERG, Thomas Jefferson University — Estimation of local organ-level blood pressure can help in diagnosing and monitoring heart and vascular diseases. Subharmonic signals from ultrasound contrast microbubbles have been proposed as a noninvasive alternative to the current practice of using manometer-tipped catheter. Approximately 10dB linear decrease in subharmonic component with 25 kPa pressure increase (typical blood pressure variation) has been reported for several contrast microbubbles. Here we report a theoretical investigation of the underlying phenomenon. We first study the well established model of a free microbubble to show that reduction of subharmonic with ambient pressure increase occurs only below a certain excitation frequency. Above this critical frequency, subharmonic signal increases with ambient pressure. Furthermore, where it decreases with ambient pressure, the relationship is linear only above certain excitation pressure. The dependence of the critical frequency on bubble radius and possibly bubble size distribution is discussed. We also report similar behavior for several models for encapsulated contrast microbubbles.

10:43AM HL.00002 Secondary flow structure from stent-induced perturbations in a bent pipe model for curved arteries, FANGJUN SHU, AUTUMN GLENN, KARTIK BULUSU, MICHAEL W. PLESNIAK, George Washington University — Secondary flow structures were investigated in a 180-degree circular bend under physiological (pulsatile) flow conditions with a stent model installed upstream of the bend. Upstream Reynolds number ranged from 200 to 1400 and the cardiac cycle period was scaled to match the physiological Womersley number, Wo=4.2. Experimental data were acquired using 2-D PIV at various cross-sectional planes along the bend. Similar to the results in absence of the stent model, symmetric counter-rotating vortex pairs were observed to develop during the cardiac cycle. In addition, transient unstable flow was initiated at the deceleration phase of the systolic peak (t/T=0.21). This complex flow is mainly attributable to perturbations induced by the stent model. It is characterized by breakdown of Dean- and Lyne-type vortices into various multiple-scale vortices. The phase-averaged flow fields were analyzed using the proper orthogonal decomposition (POD) method to gain further insight regarding the structural features of the flow.

10:56AM HL.00003 Wall shear stress as a stimulus for carotid atherosclerotic plaque progression: An MRI-based CFD pilot study, GADOR CANTON, BERNARD CHIU, TOM HATSUKAMI, WILLIAM KERWIN, CHUN YUAN, University of Washington — The aim of this study was to explore the hypothesis that intra-plaque hemorrhage, a feature associated with advanced outcomes and atherosclerotic plaque progression and destabilization, is more likely to occur in plaques with elevated levels of wall shear stress (WSS). We used multi-sequence in-vivo magnetic resonance imaging (MRI) to characterize ten human carotid atherosclerotic plaques and an MRI-based computational fluid dynamics (CFD) model to solve the equations governing the blood flow. Hemorrhage was detected within the necrotic core (intra-plaque hemorrhage) in five of these ten cases. WSS data were extracted from the results of the CFD simulations to compare patterns between the cases with and without hemorrhage. We computed the mean value of the WSS (for each time point of the cardiac cycle) at the region where a necrotic core was detected. The results from this pilot study indicate a possible link between the presence of hemorrhage within a lipid-rich necrotic core in human carotid atherosclerotic plaques and elevated levels of shear stress forcing on the luminal surface. Thus, elevated wall shear stress may be used as a high risk feature in advanced carotid atherosclerotic plaques.

11:09AM HL.00004 Mass Transport and Shear Stress in the Carotid Artery Bifurcation, RILEY GORDER, ALBERTO ALISEDA, University of Washington — The carotid artery bifurcation (CAB) is one of the leading sites for atherosclerosis, a major cause of death and disability in the developed world. The specific processes by which the complex flow found at the bifurcation and carotid sinus promotes plaque formation and growth are not fully understood. Shear stress, mass transport, and flow residence times are considered key factors. Although the governing equations closely link shear stress and mass transfer, the pulsatile, transitional, and detached flow found at the CAB can lead to differences between regions of WSS and mass transfer statistics. In this study, CAB geometries are reconstructed from patient specific 3D ultrasound medical imaging. Using ANSYS FLUENT, the fluid flow and scalar transport was solved using realistic flow conditions and various mass transfer boundary conditions. The spatial and temporal resolution was validated against the analytical solution of the Graetz-Nusselt problem with constant wall flux to ensure the scalar transport is resolved for a Peclet number up to 100,000. High residence time regions are investigated by determining the number of cardiac cycles required to flush out the carotid sinus. The correlations between regions of low WSS, high OSI, and scalar concentration are computed and interpreted in the context of atherosclerotic plaque origin and progression.
11:22AM HL.00005 Hemodynamic Simulations in Dialysis Access Fistulae\textsuperscript{1}, PATRICK MCGAH, DANIEL LEOTTA, KIRK BEACH, JAMES RILEY, ALBERTO ALISEDA, University of Washington — Arteriovenous fistulae are created surgically to provide adequate access for dialysis in patients with End-Stage Renal Disease. It has long been hypothesized that the hemodynamic and mechanical forces (such as wall shear stress, wall stretch, or flow-induced wall vibrations) constitute the primary external influence on the remodeling process. Given that nearly 50% of fistulae fail after one year, understanding fistulae hemodynamics is an important step toward improving patency in the clinic. We perform numerical simulations of the flow in patient-specific models of AV fistulae reconstructed from 3D ultrasound scans with physiologically-realistic boundary conditions also obtained from Doppler ultrasound. Comparison of the flow features in different geometries and configurations e.g. end-to-side vs. side-to-side, with the in vivo longitudinal outcomes will allow us to hypothesize which flow conditions are conducive to fistulae success or failure. The flow inertia and pulsatility in the simulations (mean $Re \approx 700$, max $Re \approx 2000$, $Wo \approx 4$) give rise to complex secondary flows and coherent vortices, further complicating the spatio-temporal variability of the wall pressure and shear stresses. Even in mature fistulae, the anastomotic regions are subjected to non-physiological shear stresses ($> 10$ Pa) which may potentially lead to complications.

\textsuperscript{1}Supported by an R21 Grant from NIDDK (DK081823)

11:35AM HL.00006 Hemodynamics in stenotic vessels: synthesis of CFD and PIV results. JENN ROSSMANN, Lafayette College — The hemodynamics in atherosclerotic blood vessels have implications for disease progression; fluid mechanical patterns and forces are linked to the risk of plaque rupture. A synthesis of numerical and experimental methods is used to investigate the dynamics in representative stenotic vessels. Detailed understanding of the hemodynamics in these vessels can contribute to prediction of rupture risk for a particular atherosclerotic plaque. Computational Fluid Dynamics (CFD) simulations of blood flow in generic and patient-specific stenotic vessels are performed using commercial software. Results of CFD are compared with those of concurrent PIV experiments to evaluate the significance of arterial wall compliance, flow pulsatility, and turbulence. Aspects of stenosis morphology are identified as useful complements to imaging modalities used in patient diagnosis and treatment.

11:48AM HL.00007 Three-dimensional Particle Image Velocimetry of Optically Opaque Flows using Ultrasound Contrast Agents. HENNING GELSHORN, UC San Diego, ANA MEDINA, UCIII Madrid, DANIEL LOTZ, DAVID J. FISHER, UC San Diego, JAVIER RODRIGUEZ-RODRIGUEZ, UCIII Madrid, JUAN C. DEL ALAMO, THILO HOELSCHER, UC San Diego — Currently, phase contrast magnetic resonance imaging is the only technique that provides time-resolved volumetric velocity maps of optically opaque systems such as blood flow in our vessels. However, this technique is expensive, time consuming and has low resolution. This project constitutes the first step towards the introduction of 3D echo-PIV, a novel ultrasound imaging technique that provides volumetric maps of three-component blood flow velocity almost in real time. 3D echo-PIV is non-invasive, fast, mobile and inexpensive, and therefore has the potential to become a commonly-used modality in the clinical setting. This new modality performs particle image velocimetry on 3D, time-lapse sequences of ultrasound bright-mode frames obtained during contrast agent infusion by tracking the sound backscattered by the contrast agent microbubbles. The present study applies 3D echo-PIV to tubular silicone phantoms in an arterial flow simulator and compares these measurements to the results from computer simulations obtained with commercial codes that have been extensively validated. We vary systematically geometrical parameters and flow rates to model different physiological hemodynamic patterns.

12:01PM HL.00008 Secondary flow structures under simple harmonic inflow in a bent pipe model for curved arteries\textsuperscript{1}, AUTUM GLENN, PENEOPE SEAGRAVE, FANGJUN SHU, KARTIK BULUSU, MICHAEL W. PLESNIKA, George Washington University — Inward centrifuging of fluid in the inviscid core of a 180 degree curved pipe leads to Lyne-type vortices under zero-mean harmonic oscillations, along with the formation of vortices in the Stokes’ layer, that rotate in the same directional sense as their steady flow counterpart (Dean vortices). Under physiological conditions, the development of the Lyne-type vortices is believed to be influenced by the systolic pulse, and its associated rapid acceleration and deceleration. Experimental data acquired using Particle Image Velocimetry (PIV) for three harmonic waveforms of different frequencies clarify the conditions under which Lyne vortices form. Multiple vortex pairs were observed for all waveforms and frequencies investigated, including Dean and Lyne-type vortex structures at a Womersley number of 4.22, much lower than previously reported. Hence, frequency alone is not an adequate governing parameter to characterize secondary flow structures in pulsatile flows. A regime map of the secondary flow was sought by using an acceleration-based parameter and the Dean number.

\textsuperscript{1}Supported by the National Science Foundation under Grant No. CBET-0909678.

12:14PM HL.00009 Relationship between potential platelet activation and LCS. SHAWN SHADDE, Illinois Institute of Technology — In the study of blood flow, emphasis is often directed at understanding shear stress at the vessel wall due to its potentially disruptive influence on the endothelium. However, it is also known that shear stress has a potent effect on platelet activation. Platelet activation is a precursor for blood clotting, which in turn is the cause of most forms of death. Since most platelets are contained in the flow domain, it is important to consider stresses acting on the platelet as they are convected. Locations of high stress can correspond to boundaries between different dynamic regions and locations of hyperbolic points in the Eulerian sense. In the computation of LCS, strain is typically considered in the Lagrangian sense. In this talk we discuss the relationship between locations of potential platelet activation due to increased stress and locations of LCS marking increase Lagrangian deformation.

Monday, November 22, 2010 10:30AM - 12:40PM – Session HM Microfluidics: General V: Particle Manipulation Long Beach Convention Center 202B

10:30AM HM.00001 Manipulating bacteria with opto-electrokinetic methods. STEVE WERELEY, JAE-SUNG KWON, SANDEEP RAVINDRANATH, JOSEPH IRUDAYARAJ, Purdue University — Recently we developed an opto-electrokinetic method for manipulating particles and cells called Rapid Electrokinetic Patterning (REP). REP is a very fast method for manipulating thousands of particles simultaneously and controllably owing to the creation of an electrothermal vortex that transports particles rapidly and in parallel to a site determined by the focal point of a laser beam. Whether particles are trapped at the center of the vortex or not is determined by their electrical properties (conductivity and permittivity). In this talk we demonstrate that REP can be used to manipulate the bacterium Shewanella oneidensis MR-1. The bacteria are assembled into large planar arrays of organisms. The dependence of this assembly process on voltage and frequency is quantified. REP can even be used to selectively manipulate and collect live or dead bacteria.
in flow cytometry, tissue engineering, and synthesis of metamaterials. Nonlinearity due to inertia can provide a platform for high-throughput passive control of particle positions in all directions, which will be useful for applications. Implementing microfluidic structures that irreversibly change interparticle spacing, similar to a low-pass filter. Although often not considered at the microscale, actions reveals a mechanism for the dynamic self-assembly process; inertial lift forces and a parabolic flow field act together to stabilize interparticle spacings into uniformly spaced lattices through purely hydrodynamic interactions with no external force fields. Numerical analysis was performed to characterize the magnetic controller. A high-speed camera provided real-time imaging of the microswimmer motion in a static fluidic environment. The robotic microswimmers exhibited active propulsion under an AC magnetic field, which demonstrates the possibility for future biomedical applications for drug delivery.

10:56AM HM.00003 Control of Biologically Inspired Robotic Microswimmers U. KEI CHEANG, Drexel University, JUN HEE LEE, Korea Institute of Machinery and Materials, DHEERAJ ROY, MIN JUN KIM, Drexel University, BAST LABORATORY TEAM — Flagella have been employed as nanoactuators for biomimetic microswimmers in low Reynolds number fluidic environments. The microswimmers utilize flagellar filaments as a constant viscous drag, anchors for microbead pinning, and a range of shear rates and are hence non-Newtonian. We model a single elastic filament in a periodic domain in both a Newtonian as well as a non-Newtonian matrix fluid. The non-Newtonian fluid model is fitted on human saliva. A body force, which is asymmetric in time, is applied to the cilium. This produces a symmetric motion of the cilium for the Newtonian case, while the motion is asymmetric for the non-Newtonian case. Due to the asymmetric motion fluid is transported in the non-Newtonian case.

11:09AM HM.00004 Ciliar fluid propulsion in a non-Newtonian liquid1, MICHEL BALTSUSSEN, PATRIC ANDERSON, Technische Universiteit Eindhoven, JAAP DEN TOONDER, Philips Applied Technologies — Natural as well as artificial cilia are used to propel fluids, or propel an animal or object through a fluid. Although the fluid is often water, other more complex fluids such as saliva and mucus are also common. These fluids are dominated by the interplay of shear-thinning viscoelastic effects, interfacial tension, and inertial lift forces. Simulations of cilia in a Newtonian fluid have received significant attention. Here, we extend these results to a non-Newtonian fluid. A focus on the dynamic self-assembly process; inertial lift forces and a parabolic flow field act together to stabilize interparticle spacings into uniformly spaced lattices through purely hydrodynamic interactions with no external force fields. Numerical analysis was performed to characterize the magnetic controller. A high-speed camera provided real-time imaging of the microswimmer motion in a static fluidic environment. The robotic microswimmers exhibited active propulsion under an AC magnetic field, which demonstrates the possibility for future biomedical applications for drug delivery.

11:22AM HM.00005 Trapping, focusing, and sorting of microparticles through bubble streaming CHENG WANG, SHREYAS JALIKOP, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Ultrasound-driven oscillating microbubbles can set up vigorous steady streaming flows around the bubbles. In contrast to previous work, we make use of the interaction between the bubble streaming and the streaming induced around mobile particles close to the bubble. Our experiment superimposes a unidirectional Poiseuille flow containing a well-mixed suspension of neutrally buoyant particles with the bubble streaming. The particle-size dependence of the particle-bubble interaction selects which particles are transported and which particles are trapped near the bubbles. The sizes selected for can be far smaller than any scale imposed by the device geometry, and the selection mechanism is purely passive. Changing the amplitude and frequency of ultrasound driving, we can further control focusing and sorting of the trapped particles, leading to the emergence of sharply defined monodisperse particle streams within a much wider channel. Optimizing parameters for focusing and sorting are presented. The technique is applicable in important fields like cell sorting and drug delivery.

11:35AM HM.00006 Controlling particle trajectories using oscillating microbubbles SHREYAS JALIKOP, CHENG WANG, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — In many applications of microfluidics and biotechnology, such as cytometry and drug delivery, it is vital to manipulate the trajectories of microparticles such as vesicles or cells. On this small scale, inertial or gravitational effects are often too weak to exploit. We propose a mechanism to selectively trap and direct particles based on their size in creeping transport flows (Re ≪ 1). We employ Rayleigh-Nyborg-Westervelt (RNW) streaming generated by an oscillating microbubble, which in turn generates a streaming flow component around the mobile particles. The result is an attractive interaction that draws the particle closer to the bubble. The impenetrability of the bubble interface destroys time-reversal symmetry and forces the particles onto either narrow trajectory bundles or well-defined closed trajectories, where they are trapped. The effect is dependent on particle size and thus allows for the passive sorting of selected sizes, on scales much smaller than the geometry of the microfluidic device. The device could eliminate the need for complicated microchannel designs with external magnetic or electric fields in applications such as particle focusing and size-based sorting.

11:48AM HM.00007 Capillary threads and droplet-decorated streams in microchannels1, SAMIRA DARVISHI, THOMAS CUBAUD, Stony Brook University — We investigate the evolution of high-viscosity fluid threads flowing in a sheath of immiscible liquid in a diverging/plane microchannel. A steady viscous-core annular flow is produced upstream in a square microchannel. Downstream, the fluids enter a diverging channel that causes the thread to bend and deform into a complex microstructure having a large interfacial area. Using a variety of fluids, we study the effect of interfacial tension, viscosities, and flow rates on the thread evolution and we characterize the evolution of the thread thickness, arc length, fold wavelength, and envelope amplitude in the plane and straight microchannel downstream from the divergence. In particular, we focus on the coalescence mechanism between adjacent folds that can produce small droplets embedded into a highly viscous matrix (i.e., “droplet-decorated” streams). Other original phenomena, such as capillary breakup by folding at high capillary numbers and secondary folding, are also examined.

12:01PM HM.00008 Inertially Stabilized Microfluidic Fluids: Self-assembly and Spatial Frequency Tuning WONHEE LEE, HAMED AMINI, UCLA, HOWARD STONE, Princeton university, DINO DI CARLO, UCLA — Dynamic self-assembly can be found over a wide range of scales originating from different fundamental opposing forces. By taking advantage of inertial effects we demonstrate controllable self-assembling particle systems at the microscale. Inertially focused particles in confined high-speed microfluidic channel flows dynamically self-assemble into uniformly spaced lattices through purely hydrodynamic interactions with no external force fields. Focusing on the dynamics of the particle-particle interaction reveals a mechanism for the dynamic self-assembly process; inertial lift forces and a parabolic flow field act together to stabilize interparticle spacings that otherwise would diverge to infinity due to viscous wakes. The interplay of the repulsive viscous interaction and inertial lift also allow us to design and implement microfluidic structures that irreversibly change interparticle spacing, similar to a low-pass filter. Although often not considered at the microscale, nonlinearity due to inertia can provide a platform for high-throughput passive control of particle positions in all directions, which will be useful for applications in flow cytometry, tissue engineering, and synthesis of metamaterials.
12:14PM HM.00009 Modification of inertial focusing position by the restriction effect , ELODIE SOLLIER, HAMED AMINI, UCLA Bioengineering, JEAN-LUC ACHARD, CNRS LEGI, DINO DI CARLO, UCLA Bioengineering, DI CARLO’S LAB TEAM, CNRS LEGI TEAM — A recent study from Faivre et al. has demonstrated that a rapid decrease in a channel cross-section leads to a lateral deviation in deformable particles’ positions downstream of the constriction. This hydrodynamic drift, or “restriction effect”, has been experimentally shown for low Reynolds numbers and only deformable cells. In parallel, Di Carlo and others have shown the usefulness of inertial forces in a confined flow, especially for ordering randomly distributed bioparticles. To further study this restriction effect but under these inertial conditions, we tracked the equilibrium positions of different bioparticles and measured their deviation. Just after the restriction, a deviation is observed for red cells and unexpectedly for spherical rigid beads. 1 cm further, beads go back to their initial position but red cells remained deviated. These results are interpreted as originating in differences in particle deformability and longer time necessary for a stressed cell to recover its initial equilibrium position. These results may provide a novel and simple technique for separation of particles with similar inertial equilibrium positions but different deformability, with specific applications for high-throughput flow cytometry.

Monday, November 22, 2010 10:30AM - 12:40PM Session HN Vortex Flows: Vortex-Induced Vibrations and Unsteady Flows Long Beach Convention Center 202C

10:30AM HN.00001 Long flexible cylinders subject to vortex-induced vibrations in shear flow exhibit preferentially counter-clockwise figure-eight trajectories at lock-in , REMI BOURGUET, Massachusetts Institute of Technology, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Long flexible cylindrical structures placed in cross-flow current exhibit large amplitude, self-excited, vortex-induced vibrations. The flow excites structure when the vortex shedding frequency is synchronized with the frequency of cross-flow vibration, a condition referred to as lock-in. When the in-line and cross-flow vibrations occur with a frequency ratio of two, as is generally observed in this context, the cylinder exhibits figure-eight trajectories. We investigate the existence of a link between the occurrence of the lock-in condition and the orientation of these figure-eight trajectories, in shear flow, by means of a joint analysis of the structure response and wake pattern, based on detailed numerical simulation and experimental results. We show that trajectories in which the cylinder moves upstream at the extremes of the cross-flow motion (counter-clockwise trajectories) are preferred by the system to establish the lock-in condition. Also, we emphasize the impact of this orientation on fluid-structure energy exchanges.

10:43AM HN.00002 Influence of flexibility and corner shape on three-dimensional vortex structures in translating plates1, DAEGYOUM KIM, MORTEZA GHARIB, California Institute of Technology — In order to understand the complicated vortex formation process of the flexible propulsors in nature, three-dimensional vortex structures generated by impulsively translating low aspect-ratio plates with a 90° angle of attack were studied experimentally. Rigid and flexible thin plastic plates were used to find the effect of plate flexibility on the development of vortex structure. The tip vortex motion is one of the obvious differences between the rigid and flexible plates. While the tip vortex moves upward in the flat-rigid plate case, it stays near the tip in the flexible plate case, which results in significantly different three-dimensional vortex morphology near the tip region. In addition, the dynamics of the vortex near a corner region was compared among three different corner angles for impulsively translating plates.

11:09AM HN.00004 Physics behind vortex-induced vibration reduction using an oblique trailing edge hydrofoil, AMIRREZA ZOBEIRI, FRANCOIS AVELLAN, MOHAMED FARHAT, Laboratory for Hydraulic Machines, EPFL, Lausanne, Switzerland — The issue of vortex-induced vibration based on the phenomenon of vortex shedding behind a bluff body is a major problem in hydraulic machinery. Resulting fluctuating forces may lead to excessive vibrations and premature cracks. It is well known that a hydrofoil with an oblique trailing edge reduces vibration as compared to that with a blunt trailing edge. However physics behind this is not fully understood. The purpose of the present work is to conduct an experimental investigation of vortex shedding dynamics in the wake of an oblique trailing edge hydrofoil to understand the phenomena and the reasons for vibration reduction. This could help optimize the trailing edge shape and diminish the induced vibration. A velocity survey in the hydrofoil wake is performed via Laser-Doppler and Particle Image velocimetry using the Proper-Orthogonal-Decomposition technique for post-processing. In addition, flow induced vibration measurements and high speed visualization are performed. The high-speed videos clearly demonstrate alternate shedding of the vortices transforming into nearly simultaneous shedding at the hydrofoil trailing edge. As a result, partial cancellation is observed for upper and lower vortices, accompanied by the thickening of the lower vortex core that is believed to be the primary reason of the vibration reduction.
present research the influence of sweeping angle on the plate tip vortex formation is studied numerically using large eddy simulation (LES). The results show an
resonance or “lock-on” phenomenon have been conducted. However, the effect of Reynolds number on the “lock-on” regime is yet to be fully understood. The
frequencies of the body, the resonant oscillations of the body can be excited, causing damaging instabilities. Various studies regarding the vortex shedding
The periodic shedding of vortices may result in significant fluctuating loading on the body. When the shedding frequency is close to one of the characteristic
stems from the vortex shedding phenomenon. It is well known that vortex streets are formed in the wake of bluff bodies over a wide range of Reynolds numbers.
The periodic shedding of vortices may result in significant fluctuating loading on the body. When the shedding frequency is close to one of the characteristic
frequencies of the body, the resonant oscillations of the body can be excited, causing damaging instabilities. Various studies regarding the vortex shedding
resonance or “lock-on” phenomenon have been conducted. However, the effect of Reynolds number on the “lock-on” regime is yet to be fully understood. The
“lock-on” phenomenon is of critical importance for the analysis of flow-induced vibrations when the aerelastic response of the structure is considered. In the
present research the influence of sweeping angle on the plate tip vortex formation is studied numerically using large eddy simulation (LES). The results show an
increase in magnitude and size of vertical structures developed in the wake of the plate.

11:48AM HN.00007 Oscillatory vortex formation behind a movable plat. MARIJA VUKICEVIC, GIANNI PEDRIZZETTI, University of Trieste — INTRODUCTION: A wide spectra of application, from industrial to environmental and biological, involve fluid-structure interaction (FSI) at a fundamental level. We investigate a 2D FSI problem for a rigid structure hinged on a wall, freely rotating by the action of an oscillatory fluid flow. METHODS: The Navier-Stokes equations are solved simultaneously with the body dynamics. An accurate numerical solution is developed on the conformational map of the time-varying physical domain. RESULTS: The FSI is primarily influenced by the vortex formation process and by the interaction between vortices generated during the sequential flow oscillations. The emerging bodies can be arranged into a three main groups. The first, made of heavy bodies, terminates the motion during the first few oscillations with the impact of the body on the wall. On the other extreme, the third group made of relatively light bodies presents a flow-driven motion that oscillates periodically in time. In a wide intermediate range, the body oscillates in time presenting non periodic features. CONCLUSIONS: The process of oscillatory vortex formation in presence of fluid-structure interaction shows the emergence of various phenomena that
were analyzed in details. In this specific application the results demonstrate that the FSI range from linear to chaotic interaction and finite-time collapse.

12:01PM HN.00008 Computational Analysis of Vortex Formation Over a Plunge Oscillating Flat Plate with Various Slip Conditions1. JOHN PALMORE, MUHAMMAD SHARIF, AMY LANG, University of Alabama — A thorough understanding of small scale aerodynamics is important for the design of micro air vehicles. Since they fly in the same Re regime as that of insects, these animals can provide biologically inspired designs. This study looks at how an alteration to the surface slip condition affects the aerodynamic flow over a wing at low Re. Butterflies have small scales (on the order of 100 microns in length) that line the surface of their wings, and it is hypothesized that these scales can affect the slip condition over their wings altering vortex formation and possibly leading to improved flight characteristics. As an initial test to this hypothesis, the flow over an infinitely thin, two-dimensional flat plate was studied using the CFD software FLUENT. The no-slip condition was modified by directly altering the fluid-slip condition over the plate. In addition, the action of flapping was simulated by varying the angle of attack as a function of time between -60 and 60 degrees. Multiple shear stress distributions, varying from shear free to no-slip, and multiple flapping frequencies were tested to discern the effects on vortex formation; lift and drag were also analyzed.

1Work performed under REU site sponsored by NSF grant EEC 0754117.

12:14PM HN.00009 Fluid flow over an elliptical cylinder undergoing a rotationally-oscillating motion, ESAM ALAWADHI, Kuwait University — The near-wake behind an elliptical cylinder undergoing rotationally-oscillating motion will be simulated using the finite element method at a low Reynolds number, Re=200. The simulations will be carried out by varying the angle of attack between ±10°, ±20°, and ±30°, while the considered range of dimensionless oscillation frequency is St*/2<St<4×Sto, where Sto is the natural Strouhal frequency of a stationary elliptical cylinder. The solver is coupled with a mesh movement scheme using the Arbitrary Lagrangian-Eulerian kinematics to simulate the flow-structural interaction. Fluid mechanics results will be presented in terms of instantaneous and time-average lift and drag coefficients, flow streamline, and vortices contours.

12:27PM HN.00010 ABSTRACT WITHDRAWN

Monday, November 22, 2010 10:30AM - 12:27PM –
Session HP Microfluids: Fluidic Devices II Long Beach Convention Center 203A

10:30AM HP.00001 Use of a porous membrane for gas bubble removal in microfluidic channels: physical mechanisms and design criteria. JIE XU, Washington State University Vancouver, REGIS VAILLANT, DANIEL ATTINGER, Columbia University — We demonstrate and explain a simple and efficient way to remove gas bubbles from microchannels, by integrating a hydrophobic porous membrane on top of the microchannel. A prototype chip is made in PMMA with the ability to completely filter gas plugs out of a segmented flow at rates up to 7.4 µL/s/mm². In our device, gas plugs in a water stream are generated continuously from a T-junction and are then transported towards the gas removal section, where they slide along and vent through a hydrophobic membrane. To achieve complete gas removal without membrane leakage, our analysis shows that four necessary operating criteria are needed. These criteria are verified by experimental results. The first criterion is that the bubble length needs to be larger than the channel diameter. The second criterion is that the bubble should stay on the membrane for a time sufficient to transport all the gas through the membrane. The third criterion is that the bubble travel speed should be lower than a critical value: otherwise a stable liquid film between the bubble and the membrane prevents mass transfer. The fourth criterion is that the pressure difference across the membrane should not be larger than the Laplace pressure to prevent water from leaking through the membrane. Experiments on our device show a good agreement with these criteria.
10:43AM HP.00002 High-speed µ-PTV study of microbubble generation in microfluidic T-junction. RYOJI MIYAZAKI, TOSHIYUKI OGASAWARA, MITSUHISA ICHIYANAGI, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — The bubble generation in a microfluidic T-junction is investigated by high-speed imaging to develop a novel technique for monodispersed microbubble generation. The proposed technique enables generation of 20 ~ 70 µm diameter bubbles at frequency of 1 ~ 10^2 kHz, under the mean liquid velocity at the order of 1 m/s. The generation process is quantitatively analyzed focusing the time change of the gas area and the distance between the receding interface and the channel corner. The µ-PTV (micron-resolution Particle Tracking Velocimetry) is operated to measure the flow field on the bubble generation by seeding 1.0 µm particles with bright-field microscopy. The bubble generation process is highly periodic; therefore, µ-PTV is iteratively conducted in the same phase of the bubble generation. Time-series velocity-vectors at the order of 1 m/s are measured by this high-speed µ-PTV method. The high-speed imaging indicates that the bubble generation consists of two stages: intruding stage and squeezing stage. The terminal gas area is largely determined by the gas area at the beginning of the squeezing stage. According to the obtained flow field, the liquid rapidly flows into the side channel with the growth of the gas tip.

10:56AM HP.00003 Reaction Kinetics in Micro/Nanofluidic Devices: Effect of Confinement and AC Voltage. VISHAL V.R. NANDIGANA, NARAYANA R. ALURU, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Owing to limited sample consumption, electrokinetic control of convective transport and rapid dissipation of heat, nanofluidic devices are currently being investigated extensively in the field of chemical reactions. The reactants are typically transported into the nanochannel by using external DC electric fields. In this study, a novel technique to increase the rate of catalytic reactions inside nanofluidic devices is presented. Specifically, the effect of combined AC and DC electric fields on different reaction kinetics was numerically investigated and it was found to enhance the rate of formation of desired species in reaction limited kinetics (when the Damköhler number (Da) ≤ 1). We investigate the role of AC frequency, amplitude, channel height and surface charge density on reaction kinetics. We develop analytical expressions for fluid transport under combined AC and DC fields and also develop expressions to identify optimal frequencies. Several examples are considered to illustrate the effect of AC fields on chemical reactions in nanochannels.

11:09AM HP.00004 Melt Crystallization in Microfluidics for Sample Concentration. POORIA SHARIF-KASHANI, H. PIROUZ KAVEHPOUR, University of California, Los Angeles — Melt crystallization in microfluidics is a novel approach to concentrate/purify a diverse range of samples from particles to ions. In this technique, the difference in solubility of solutes in the liquid and solid phase of the solvent drives the transport of the solutes. Consequently, this method has the advantage of being non-invasive and entirely thermally-actuated with no moving parts. A fluid sample is frozen in a microchannel and melting zones are passed repeatedly through the stationary sample to increase the concentration of solute at one end. The device is constructed using a thermoelectric cooler to freeze the sample and thin-film resistive heaters to create melting zones. The heaters are operated independently, allowing them to be switched on or off to create a localized melting zone in the channel. The performance of the system is successfully tested for a variety of samples including aqueous solutions and water containing micro-particles.

11:22AM HP.00005 Charge Transport Behavior in Microfluidic Microbial Energy Conversion Devices. ALOKE KUMAR, PARTHA MUKHERJEE, ABHJEET BOROLE, MITCHEL DOKTYCZ, Oak Ridge National Laboratory, Oak Ridge, TN 37831, USA — Microbial energy harvesting devices utilize anode-respiring bacteria (ARB), present as a biofilm matrix, to generate electrical current from organic matter. The conductive biofilm matrix in the anode compartment plays a key role in the overall charge transport behavior. Especially, biofilm kinetics and ARB community dynamics are of paramount importance influencing the anode overpotential, which is further dependent on the pH variation. In this work, we present a theoretical framework to study the charge transport characteristics with concomitant biofilm kinetics, substrate utilization, diffusion and migration in a microfluidic device with microbial energy generation.

11:35AM HP.00006 Transporting Microparticles Using a Conveyor Belt of Artificial Cilia. AMITABH BHATTACHARYA, Department of Chemical Engineering, University of Pittsburgh, GAVIN BUXTON, Department of Science, Robert Morris University, ALEXANDER ALEXEEV, Department of Mechanical Engineering, Georgia Institute of Technology, O. BERK USTA, Harvard Medical School, ANNA C. BALAZS, Department of Chemical Engineering, University of Pittsburgh — We present results from simulations of particle transport in a fluid microchannel via a regular array of actuated cilia. For each cilium, one end is tethered to the wall, while the other end is actuated by an external periodic force. This leads to a time-asymmetric, cyclic motion for each cilium. We study the motion of a microparticle in the fluid due to the cilia actuation. An adhesive force between the particle and cilia enables a transport mechanism for the particle in which the particle is passed from one cilium to the next cilium in the array. The particle is also dragged forward by flow in the channel, induced by the time-asymmetric motion of the cilia. The simulations are performed using the Lattice Boltzmann Method for the flow, with a chain of point-forces, connected by springs, used to represent each cilium. We will present the parameter regime where the most effective transport of the particle occurs due to the combination of cilia-particle adhesion and fluid motion.

11:48AM HP.00007 Induced-Charge Electro-Osmosis Micropumps for Portable Microfluidics: theory and experiment. JOEL PAUSTIAN, TODD SQUIRES, UCSC Chemical Engineering — Microfluidic devices (e.g. Labs on a Chip) are becoming useful scientific and medical tools for automating chemical and biological lab work. Various impediments prevent complex microfluidic devices from being easily removed from a laboratory setting, limiting their utility for day-to-day applications like in-the-field medical diagnostics and drug delivery. The development of portable and integrable high-pressure pumping techniques will be necessary step for truly portable, complex microfluidic devices. Microfluidic pumps based on the electrokinetic phenomenon of Induced-Charge Electro-Osmosis (ICEO) could potentially fill this role. We investigate the role of AC frequency, amplitude, channel height and surface charge density on reaction kinetics. We develop analytical expressions for fluid transport under combined AC and DC fields and also develop expressions to identify optimal frequencies. Several examples are considered to illustrate the effect of AC fields on chemical reactions in nanochannels.

12:01PM HP.00008 Regulating flow with substrate shape in capillary micropumps. MATTHEW HANCOCK, Brigham & Women’s Hospital, JOHN BUSH, MIT — Capillarity offers a passive mechanism to pump fluid through portable lab-on-a-chip systems, making them ideal for rapid in situ analysis of medical samples in the developing world. A common capillary micropump design is powered by the difference in curvature pressures between drops at the inlet and outlet of a microchannel. The resulting flow rate is transient, depending on the geometry of the inlet cavity and the instantaneous droplet volumes. We here present a class of microcavity shapes that maintain constant pressure within droplets regardless of their volumes. This special class of microcavities may prove useful for regulating pressure in microfluidic devices. We suggest the design of a passive capillary micropump fitted with a special pressure regulating inlet cavity that forces a constant flux through a microchannel. The influence of gravity on this class of microcavities is considered.
12:14PM HP.00009 Pumping of Dielectric Liquids Using Non-Uniform-Field Induced Electrohydrodynamic Flow1. JAE CHUN RYU, WONKYOUNG KIM, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — Pumping of dielectric liquids or poorly conducting liquids is necessary in cooling of microelectronic devices, dispensing liquids in miniature systems for chemical and biological analysis, and micropumping of organic solvents for microreactor. Electrical pumping of liquids is more attractive than conventional mechanical pumping methods because of many advantages such as simple design, no mechanical parts, low acoustic noise, and lightweight. We present a new electrohydrodynamic (EHD) pumping method for dielectric liquids. The pumping method relies on the EHD flow generated by electric-field dependent electrical conductivity (Onsager effect). A polar additive plays an important role in enhancing the field-dependency of conductivity. When ac voltage is applied, a fast and regular flow was produced around electrodes. Flow speed is proportional to cube of electric-field strength and inversely to applied frequency. The experimental results showed good agreement with theoretical analysis which is based on our model.

1 This research was financially supported by a grant to MEMS Research Center for National Defense funded by Defense Acquisition Program Administration.

Monday, November 22, 2010 10:30AM - 12:40PM Session HQ Bubbles II Long Beach Convention Center 203B

10:30AM HQ.00001 Tip vortex cavitation suppression by water ejection from wing tip. MOHAMED FARHAT, MARTINO RECLARI, Laboratory for Hydraulic Machines - EPFL — In the present study we investigated how a water jet, used to create a winglet-like effect, actively reduces or suppress the cavitation formed into the core of a tip vortex. Modifications of the vortex structure were monitored by measuring the velocity profiles with laser Doppler velocimetry. High-speed jets proved to be very effective in increasing the size of the vortex core, thus inhibiting the formation of tip vortex cavitation.

10:43AM HQ.00002 Tip vortex cavitation suppression by mass injection. HARISH GANESH, University of Michigan, Ann Arbor, NATASHA CHANG, Naval Surface Warfare Center, Carderock Division, STEVEN CECCIO, University of Michigan, Ann Arbor — Injection of water and aqueous polymer solutions into the core of a trailing vortex is found to delay the onset of tip vortex cavitation (TVC). For the case without any mass injection, cavitation inception ($\sigma_1 = 3.3$) occurred at a substantially higher pressure ($C_{p_{min}} = 2.3$) than that was expected based on the mean vortical flow. Mass injection (both water and polymer) into the vortex core led to a reduction in the inception pressure. 2-D Particle Image Velocimetry was performed in a region of flow in the vicinity of the average inception location near the hydrofoil tip to determine the instantaneous flow fields near the vortex core. Mass injection led to significant modification of the unsteady flow field, while the average flow field was not strongly affected. A 50% reduction in RMS velocities with comparison to the non-injection conditions was observed for the case of polymer injection, the case of maximum cavitation suppression. Cavitation inception/desinception studies were conducted for different mass and momentum fluxes to develop a hypothesis for TVC suppression.

10:56AM HQ.00003 Inviscid Partial Coalescence from Bubbles to Drops. F.H. ZHANG, Singapore-MIT Alliance, National University of Singapore, Singapore, P. TABOREK, Department of Physics and Astronomy, University of California, Irvine, California, USA, J. BURTON, James Franck Institute, University of Chicago, Chicago, Illinois, USA, B.C. KHOO, K.M. LIIM, Singapore-MIT Alliance, National University of Singapore, Singapore, S.T. THORODDSEN, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia — Coalescence of bubbles (drops) not only coarsen the bubble (drop) sizes, but sometimes produces satellite bubbles (droplets), known as partial coalescence. To explore links between the drop and bubble cases, we experimentally study the partial coalescence of pressurized xenon gas bubbles in nano-de-ionized water using high-speed video imaging. The size of these satellites relative to their mother bubbles is found to increase with the density ratio of the gas to the liquid. Moreover, sub-satellite bubbles are sometimes observed, whose size is also found to increase with the density ratio, while keeps about one quarter of the primary satellite. The time duration from start of the coalescence to formation of the satellites, scaled by the capillary time, increases with the density ratio too. In addition, as the size ratio of the father bubble to the mother bubble increases moderately, their coalescence proceeds faster and the sub-satellite is prone to form and relatively larger.

11:09AM HQ.00004 Contact time of a pair of bubbles in an acoustic field. MINORI SHIROTA, Hiroaki University, HIROI MIYAMAE — Contact time of a pair of bubbles in an acoustic field is investigated experimentally. Pairs of bubbles of about 0.1 mm in radius were exposed in an acoustic field of about 30 kHz. The bubbles were generated near T-shaped microfluidic junction in silicone oil of 50 cSt with actively controlled gas pressure change. These bubbles were then introduced in an acoustic levator commonly used in single bubble coalescence experiment. The contact time of the oscillating bubbles were quantitatively evaluated using high-speed imaging technique. Bubbles with in-phase volume oscillation attracted each other due to secondary Bjerknes force and finally coalesced. We observed bubbles smaller than resonant sizes at large separate distance deforms greatly and coalesced immediately when they touch, while bubbles of the resonant sizes were kept contacting such long time as over 100 periods of forcing.

11:22AM HQ.00005 Collisions between a rising bubble and a rigid sphere. A. BELMONTE, W. G. Pritchard Labs, Dept of Mathematics, Penn State, S.T. THORODDSEN, Division of Physical Sciences and Engineering, KAUST, Saudi Arabia — Motivated by studies of particle sedimentation in bubbly turbulence, we perform an experimental study of the controlled collision between a rising bubble and a rigid sphere. Both stationary and sinking spheres are considered. Impact dynamics including bouncing are measured as a function of relative sizes, collision velocity, buoyancy, surface tension, and the viscosity of the continuous phase. High-speed video imaging is used to measure the bubble deformations and the induced capillary waves. We estimate the effective coefficient of restitution of the collision and compare to recent models for bubbly sedimentation.

11:35AM HQ.00006 The rupture dynamics of ultra-viscous bubbles. JAMES BIRD, Massachusetts Institute of Technology, HOWARD STONE, Princeton University, JOHN BUSH, Massachusetts Institute of Technology — 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, the rupture dynamics are dominated by surface tension and inertia and typically occur over a period of milliseconds. In liquids with relatively high viscosity, the viscous dissipation slows the dynamics enough that gravity, rather than surface tension, is believed to be responsible for the bubble collapse. However, here we demonstrate that gravity is responsible for neither the collapse nor the resulting instability that wrinkles the film. Using a combination of experiments and theory, we investigate why capillary forces display attributes that are normally exclusive to gravitational forces.

11:48AM HQ.00007 Singular jets in the formation of bubbles in viscous fluids. THOMAS SEON, VIRGINIE DUCLAUX, ARNAUD ANTKOWIAK, CNRS & UPMC Univ Paris 06, Institut Jean Le Rond d’Alembert, Paris, France — We study experimentally the process of formation of large bubbles in viscous fluids. Whereas at low flow rates, the produced individual bubbles quickly recover a quasi-spherical shape, collective behaviors between bubbles are identified as the feeding gas flow rate is increased. These interactions may lead to the surprising gobbling of a bubble by another, resulting in large sized bubbles with inner viscous shells. At even higher feeding rates, a violent Worthington jet following bubble pinch-off appears. This jet is so intense and concentrated that perforation of the bubble may occur. We may analyze the whole phenomenology of the large interface deformations associated with bubble formation in viscous fluids with detailed experiments conducted with high-speed video imaging.
12:01PM HQ.00008 Marginal Pinching in a Bubble Sitting on a Solid Surface, Guillaume Berteloot, Piouz Kavehpour, Pooriah Sharif-Kashani, UCLA — While the shape of a bubble is widely understood to be dictated by a competition between film elasticity and Laplace pressure induced by the curvature of the latter, the shape of the liquid film in contact with the solid surface has not been studied. Using fluorescence microscopy techniques, we show that the inner liquid/air interface exhibits a dip. This can be related to marginal pinching, and simulations show good agreement between theory and experiment. This dip depends on the surfactant used for bubble formation, and the height difference increases with time. This feature can be of importance, because bubbles can be seen as a base unit for foams, which are widely used for medical as well as industrial purposes, such as enhanced oil recovery. For those applications, the interaction between the foam and the substrate is crucial, as the quality of the foam hence the efficiency of the process depends on it. The study of foams can be applied to bubbles as one can see the vicinity of the contact line as part of a Plateau border.

12:14PM HQ.00009 Explosion-Induced Implosions of Cylindrical Shell Structures1, C.M. Ikeda, J.H. Duncan, University of Maryland — An experimental study of the explosion-induced implosion of cylindrical shell structures in a high-pressure water environment was performed. The shell structures are filled with air at atmospheric pressure and are placed in a large water-filled pressure vessel. The vessel is then pressurized to various levels $P_c = \alpha P_L$ where $P_c$ is the natural implosion pressure of the model and $\alpha$ is a factor that ranges from 0.1 to 0.9. An explosive is then set off at various standoff distances, $d$, from the model center line, where $d$ varies from $R$ to $10R$ and $R$ is the maximum radius of the explosion bubble. High-speed photography (27,000 fps) was used to observe the explosion and resulting shell structure implosion. High-frequency underwater blast sensors recorded dynamic pressure waves at 6 positions. The cylindrical models were made from aluminum (diameter $D = 39.1$ mm, wall thickness $t = 0.89$ mm, length $L = 240$ mm) and brass ($D = 16.7$ mm, $t = 0.36$ mm, $L = 152$ mm) tubes. The pressure records are interpreted in light of the high-speed movies. It is found that the implosion is induced by two mechanisms: the shockwave generated by the explosion and the jet formed during the explosion-bubble collapse. Whether an implosion is caused by the shockwave or the jet depends on the maximum bubble diameter and the standoff distance.

1The support of the Office of Naval Research is gratefully acknowledged.

12:27PM HQ.00010 Single cavitation bubble dynamics in micro-channels near free and rigid boundaries, Oscar Enriquez, Devaraj van der Meer, Detlef Lohse, University of Twente, Claus-Dieter Ohl, Nanyang Technological University — It is well known that cavitation bubbles jet towards a rigid interface and away from a free surface. Yet, cavitation bubbles between a free and a rigid boundary show more complex deformation and the direction of jetting depends on a delicate interplay of attractive and repulsive forces. We re-investigate this regime in the context of microfluidics. We use laser-induced cavitation bubbles and high-speed photography to study their dynamics. The cavitation bubble is located between two channel walls (500 $\mu$m apart) and a free surface. We vary the distance of both the free interface and the bubble from the walls. In most of the parameter space we observe the expected dynamics. Yet, between these scenarios we find jetting directed towards the liquid-air interface as well as axisymmetric collapse without a jet. Additionally, we find complex dynamics of the free interface.

Monday, November 22, 2010 10:30AM - 12:40PM — Session HR Drops VIII: Impact Long Beach Convention Center 203C

10:30AM HR.00001 Instant capillary origami, Marco Rivetti, Basile Audoly, Sebastien Neukirch, Christophe Josserand, Arnaud Antkowiak, Institut D'Alembert, Paris, CNRS & UPMC — A liquid drop impacting a thin elastic membrane forms a ‘dynamical capillary origami’ on the very rapid capillary timescale. Dynamics is here a key ingredient that allows for shape selection of the elasto-capillary bundle based only on the impact velocity. We study this phenomenon using a simplified 2D setup, where a drop impacts a narrow polymer strip. This experiment exhibits a surprisingly rich variety of phenomena: coupled capillary and elastic waves, drop breakup, jet ejection… We propose a very simple model that mimics the complex fluid-structure interactions at play. The results of this model are in close agreement with the full experiment, both in the qualitative dynamics of folding and in the quantitative representation of the phase diagram of encapsulation. Eventually, we discuss how drop dynamics may induce anomalous encapsulation events where unexpectedly long strips are wrapped around drops.

1Support from ANR Deformation ANR-09-JCJC-0022-01.

10:43AM HR.00002 Impact of Microdrops on Solids: Modelling and Simulation, James Sprittles, Yuli Shikhmurzaev, University of Birmingham — A major obstacle to the design of ink-jet printing devices for targeted deposition of microdrops is that drop dynamics may induce anomalous encapsulation events where unexpectedly long strips are wrapped around drops.

10:56AM HR.00003 Droplet Impact on Inclined, Planar Surfaces, G. Paul Neitzel, Phares Carroll, Georgia Institute of Technology — The impact of a liquid droplet on a planar surface is of interest in a variety of applications ranging from droplet-impingement cooling to forensic blood-spatter analysis. An experimental system capable of generating liquid droplets of varying diameters and velocities of relevance to the latter of these applications has been developed for use in an educational context by secondary-school students. Experiments have been performed to quantify droplet patterns corresponding to several relevant dimensionless parameters, i.e., the Weber number, contact angle, impact/inclination angle, and roughness ratio. Results show that characteristics of droplet collisions, namely the eccentricity of the splash zone and creation of spines from a droplet’s corona, can be attributed to and predicted by these dimensionless parameters for the range of inclination angle, Weber number, and impact surfaces included in the present study.

1Supported by NASA and NSF.
An experimental study of Mesler entrainment in silicone oil. J.R. SAYLOR, G.D. BOUNDS, Clemson University — When a drop impacts a flat liquid surface, bubbles can be entrained into the bulk. At relatively low impact velocities, a large number of very small bubbles can form, a process typically referred to as Mesler entrainment. Virtually all studies of Mesler entrainment have utilized water as the working fluid. Water surfaces are notoriously difficult to maintain in a clean state and the possible effect of contaminating monolayers has clouded the extant work. In the present study, silicone oil, which is significantly less susceptible to surface contamination, was used as the working fluid, allowing us to determine the effect of contaminants. The experimental conditions were adjusted so that the range of Weber, Froude, and Capillary numbers \((We, Fr, Ca)\) were comparable to the general range investigated in water-based studies. Several differences were observed between the silicone oil results and those for water. Of especial interest is that Mesler entrainment tends to occur either all of the time, or none of the time in silicone oil while, for water, Mesler entrainment occurs intermittently. Experiments were also conducted with a more viscous silicone oil. Here, \(Ca\) was increased by \(\sim 100\) times while keeping the same general \((We, Fr)\) range. Significant differences were observed in the \((We, Fr)\) location where Mesler entrainment occurred, when compared to the lower viscosity silicone oil runs. These results are believed to be the first to reveal the effect of \(Ca\) on Mesler entrainment.

Singularity and jet formation for drop impact on a dry surface. LAURENT DUCHEMIN, IRPHE, CNRS & Aix-Marseille Univ., CHRISTOPHE JOSSERAND, Institut D’Alembert, Paris. CNRS & UPMC — We study the influence of the surrounding gas in the dynamics of drop impact on a smooth surface. We use a model for which both the gas and the liquid are incompressible; lubrication regime applies for the gas film dynamics and the liquid viscosity is neglected. We show that in the absence of surface tension a singularity is formed when the liquid touches the solid, while a thin film of air always remains present between the solid and the liquid when surface tension is taken into account. We explain the self-similar structure of the singularity and we show that the jet thickness is proportional to the capillary length of the impact.

On air entrapment and splashing threshold in drop impacts. CHRISTOPHE JOSSERAND, PASCAL RAY, STEPHANE ZALESKI, Institut D’Alembert, Paris. CNRS & UPMC — We investigate here how the surrounding gas influence the dynamics of drop impacts on a thin liquid film. We describe in details the entrapment of the gas bubble using numerical simulations with high enough mesh resolution. The bubble entrapment comes from viscous effect in the thin gas layer that need to be evacuated down the drop, creating a high pressure field that deforms the drop interface into a dimple. We finally investigate how this dynamics coupling gas and liquid dynamics can change the splashing dynamics.

Direct Numerical Simulation of Cell Printing. RUI QIAO, PING HE, Clemson University — Structural cell printing, i.e., printing three dimensional (3D) structures of cells held in a tissue matrix, is gaining significant attention in the biomedical community. The key idea is to use desktop printer or similar devices to print cells into 3D patterns with a resolution comparable to the size of mammalian cells, similar to that in living organs. Achieving such a resolution in vitro can lead to breakthroughs in areas such as organ transplantation and understanding of cell-cell interactions in truly 3D spaces. Although the feasibility of cell printing has been demonstrated in the recent years, the printing resolution and cell viability remain to be improved. In this work, we investigate one of the unit operations in cell printing, namely, the impact of a cell-laden droplet into a pool of highly viscous liquids using direct numerical simulations. The dynamics of droplet impact (e.g., crater formation and droplet spreading and penetration) and the evolution of cell shape and internal stress are quantified in details.

Impact and bouncing of a liquid onto an inclined wet surface. TRISTAN GILET, JOHN BUSH, MIT — We report the results of an experimental investigation of the impact of droplets onto a solid planar surface coated with a thin layer of high viscosity silicon oil. Particular attention is given to deducing criteria for bouncing, and elucidating the energetics of impact. The viscosity, size and impact velocity of the droplet are varied, as well as the inclination of the surface. The motion is recorded with a high speed camera and the energy transfers are measured by image processing. The principle dissipation mechanisms are discussed, and scaling laws proposed for the parameters characterizing the impact (e.g. coefficient of restitution, contact time, slip length). Our results are compared to those reported in previous studies of bouncing.

Dynamics of the ejecta sheets generated by a drop impact. MARIE-JEAN THORAVAAL, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology, Thuwal 23955-6900, Saudi Arabia — At high Reynolds and Weber numbers, a drop impacting on a liquid layer produces a thin ejecta sheet between the drop and the pool. Ultra-high speed imaging reveals the complex ejecta shapes produced by the impact. We have characterized the evolution of the sheet for different viscosities (mixtures of water and glycerin) and impact velocities. It appears that the relevant parameters are the Ohnesorge number and the splash parameter. For high splash parameters, the ejecta sheet bends toward the pool until it impacts the surface, trapping some air. Then a slingshot mechanism ejects droplets at high velocities from the external part of the sheet. Micro-droplet velocities suggest that the sheet becomes as thin as 100 nm. At intermediate Ohnesorge numbers, some bumps disturb the regularity of the sheet. More complex evolutions can be observed, including folding of the sheet, self-intersecting sheets, waves propagating in the sheet and explosive breakup of the sheet.

Rosace patterns in drop impact. GUILLAUME LAGUebaU, LAUM, UMR CNRS 6613, Le Mans France, MARCO FONTELOS, Instituto de Ciencias Matematicas, Madrid, CHRISTOPHE JOSSERAND, Institut D’Alembert, UMR 6613, UPMC Paris, AGNES MAUREL, Institut Lavoisier, UMR 7687, ESPCI Paris, VINCENT PAGNEUX, LAUM, UMR 6613, Le Mans France, PHILIPPE PETITJEANS, PMMH, UMR 7636, ESPCI, France, ANR-08-BLAN-0108 TEAM, ANR-09-JCJC-0022 COLLABORATION, MTT-2008-03255 COLLABORATION — We report an experimental study of the instability of the corolla for drop impacts on liquid surface for moderate Weber numbers \((We)\) and millimetric liquid layers (of thickness \(h\)), where no splash is observed. Thanks to a Fourier Transform Profilometry technique (FTP), we exhibit and analyze for the first time the formation of a rosace-like pattern originated from an hydrodynamic instability. Using the shallow water approximation, we explain the main mechanisms leading to these patterns: it consists in the linear instability of the self-similar axisymmetric radial solution of the equations. We found that the number of folds scales like \(We/h\) at the power 2/7 as observed in our experiments.

Monday, November 22, 2010 10:30AM - 12:27PM – Session H5 Drops IX: Impact Long Beach Convention Center Grand Ballroom A
10:30AM HS.00001 Beneath a splash: interference imaging of the air below a spreading drop
MICHHELLE DRISCOLL, SIDNEY NAGEL, The James Franck Institute, University of Chicago — Viscous splashing produces a distinct morphology: an impacting drop spreads as a thick lamella, which at late times ejects a thin liquid sheet that subsequently breaks into drops. We describe an interference technique to measure the air layer beneath the spreading drop using high-speed, high-resolution video imaging. We use this technique to visualize the dynamics of the entrapped air above a drop. In addition, we measure the air gap between the thin sheet and the substrate, and find that, within experimental resolution, there is no air gap beneath the spreading lamella. At the lamella/substrate interface we find a surprising feature: small regions of the expanding thin sheet is approximately 1 μm above the substrate, frequently touch down and wet the surface. Pockets of air become trapped among these contacts, and form air bubbles. Air entrainment occurs only when the thin sheet is present; when it is suppressed the lamella spreads as a smooth wetting front.

10:43AM HS.00002 Experimental and theoretical study of pattern identification in physical systems on circular domains
RORY HARTONG-REDDEN, Northwestern University, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — This study is motivated by a recent finding of systems in the context of fluid dynamics (e.g. drop splash phenomena) where instabilities of different wavenumbers may co-exist and thus lead to several single-wavenumber patterns superimposed with random phase-shifts between them. A new experimental setup of stereo high-speed photography enables accurate data acquisition of these patterns. However, identification of the wavenumber structure of such patterns from experimental data is complicated by the lack of a theoretical basis as well as the presence of experimental uncertainties and possible missing points in the data. We present both a new theoretical framework and an example of application — the crown structure analysis in the drop splash problem.

10:56AM HS.00003 The Generation of Secondary Droplets due to Drop Impact on a Water Surface in the Presence of Wind and Surfactants
REN LIU, XINAN LIU, University of Maryland — The impact of single water drops on a water surface was studied experimentally in a wind tunnel. The wind speed ranged from 0 to 10.0 m/s. After leaving the needle, the drops move downward due to gravity and downstream due to the effect of the wind, and eventually hit a pool of water on the bottom of the test section. The drop impacts were recorded simultaneously from the side and above with two high-speed movie cameras set at 1,000 frames per second. It is shown that the water drop obliquely impacts the water surface and the impingement angle relative to vertical increases with increasing wind speed. After the drop hits the water surface, a chain of secondary droplets is formed and moves in the leeward direction. This is followed by a stall formation at the location of drop water impact. The effects of wind speed and initial drop size on the number, diameter and total mass of secondary droplets and the shape of the stall were investigated. The effects of surfactants on these parameters were also studied.

1The support of the National Science Foundation under grant ARCO962107 is gratefully acknowledged.

11:09AM HS.00004 Qualitative Observations of Droplet Impact on Superhydrophobic Surfaces with Micro-ribs for Three Fluids
JOHN PEARSON, DANIEL MAYNES, BRENT W. WEBB — Droplet impingement experiments on superhydrophobic surfaces with micro-ribs and a hydrophobic coating were performed using three fluid types: water, a 50/50 water/glycerine mixture, and ethanol. Also tested, for comparison, were patterned and uncoated, smooth coated, and smooth uncoated surfaces. For surfaces with rib and cavity features, the droplet spread and retraction were observed to be asymmetric and at high Weber numbers the spread along the rib direction is greater than the spread transverse to the ribs. The onset of peripheral spashing was observed to be contact angle dependent and preferential along the ribs. The occurrence of two-pronged and oscillating jets during droplet retraction for water/glycerine tests was observed when the surface was micropatterned. The oscillating and splitting jets were stronger on the superhydrophobic surfaces than on the surfaces with patterning but no hydrophobic coating. Further, an interesting spread pattern with four liquid droplets clustered at about 30° from the perpendicular direction was observed for all fluid types on both coated and uncoated patterned surfaces.

11:22AM HS.00005 Analogies between a drop impacting a solid surface, an oscillating sessile drop, and two coalescing drops
SANTOSH APPATHURAI, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — Numerous industrial processes involve the formation of drops which then collide with a solid substrate or another drop. When a drop impacts a solid surface without splashing, it may spread on the surface and then recoil, much like an oscillating sessile drop whose contact line is free to move. Were the impacting drop to make a fixed contact angle of 90 degrees with a substrate that exerts negligible viscous drag on the drop, the situation approximates well the aftermath of what happens once two drops have just started coalescing. Given the aforementioned analogies between these apparently quite distinct physical problems, the dynamics of each process is analyzed by solving numerically the 3D axisymmetric or 2D Navier-Stokes system using a well-benchmarked ALE algorithm based on the Galerkin/Finite Element Method (G/FEM) for spatial discretization and adaptive finite differences for time integration.

11:35AM HS.00006 Capillary waves on a periodically supported liquid cylinder in low gravity
DAVID THIESSEN, FAHIM CHANDURWALA, LIKUN ZHANG, Washington State University — The impact of a droplet on a capillary channel consisting of a helical wire filled with water generates capillary wave packets that propagate away from the impact zone helping to dissipate the droplet kinetic energy. A simplified model is presented for the channel consisting of a periodic array of wire rings that penetrate the surface of the liquid. The multiple scattering problem for monochromatic capillary waves on a liquid cylinder impinging on an array of concentric rings is solved by the finite element method (FEM) with radiation boundary conditions at each end. Contact lines are taken to be pinned on the wires. In the limit of wires of infinitesimal thickness the FEM results agree with a semi-analytical theory. The results also allow the determination of an effective wave speed. Finally, results of experimental measurements of wave-packet speed on small horizontal capillary channels will be discussed.

1Supported by NASA.

11:48AM HS.00007 Thermal Boundary Layer Dynamics in Multiple Droplet Impingement
MARIO F. TRUJILLO, STEVEN LEWIS, EELCO GEHRING, University of Wisconsin — The impingement of a stream of HFE-7100 droplets striking a pre-wetted and heated surface is studied for droplet Weber and Reynolds numbers ranging from 285 to 427 and 1250 to 4850, respectively, and for a film depth to droplet diameter ratio varying from 0.4 to 1.5. After a short period, a quasi-steady state is achieved; in which the liquid crown formed during continuous droplet impact remains nearly stationary. Temporal averages of the velocity, temperature, and liquid fraction fields suggest that the boundary layer can be categorized as consisting of a stagnation point flow region, a linear growth section, and a jump region, similar to a hydraulic jump, near the liquid crown. Results of the average radial temperatures are compared to experiments for various heat fluxes yielding good agreement. Additionally, it is shown that a sub-layer is present in all cases considered, which is categorized by low values of the local Peclet and Reynolds numbers. The heat transfer mode in this sub-layer domain is governed to a great degree by conduction, and experiences a delayed cooling effect.

1The authors are grateful for the support from the Office of Naval Research, code 331, and to Mark Spector, its program director.
12:01PM HS.00008 Viscous drop collisions on surfaces of varying wettability. DANIEL BOLLEDUULA, University of Washington, AL BERCHIELLI, Pfizer Inc., ALBERTO ALISEDA, University of Washington — We present an experimental study of increasingly viscous acetone rich and Newtonian equivalent liquid drops colliding on surfaces of varying wettability. This class of liquids applies directly to spray coating processes in pharmaceutical industries. The results from this study will elucidate the physics in a regime where resisting viscous forces and the restoring forces of capillarity are balanced, \( Oh \sim 1 \). Early spreading dynamics \( \tau = Ut/D \ll 1 \) indicate negligible dependence on contact angles while longer times demonstrate deviations from Tanner’s law, \( D \sim t^{1/10} \). We will compare our results with recent theory to demonstrate the feasibility of modelling complex rheology spreading characteristics over short and long time scales. Preliminary results indicate an intermediate spreading regime following the inertial phase where the diameter, \( D \sim t^n \) with \( 1/7 < n < 1/5 \).

12:14PM HS.00009 Drop fragmentation due to hole nucleation during Leidenfrost impact. CHRISTOPHE PIRAT, ANNE-LAURE BIANÈCE, CHRISTOPHE YBERT, LPMCN, Université Lyon 1 et CNRS — Drop impact on a smooth plate heated above the Leidenfrost temperature is investigated in the range of large Weber number. Liquid fragmentation due to the rupture of the expanding lamella during the impact is studied experimentally. This rupture can be triggered by the presence of a small defect on the surface, which acts as a nucleation site for the hole formation, whereas the liquid does not contact the substrate. The rupture is shown to take place above a critical impact velocity, the lower when the defect size is the larger. This mechanism of rupture is compared to classical splash. It is shown to be relevant if the drop size \( R_0 \) and the size of the defect \( d \) are below a critical ratio \( R_0/d \leq 40 \).

Monday, November 22, 2010 10:30AM - 12:40PM
Session HT Bioluminiscence VI: Flapping and Flying II Long Beach Convention Center Grand Ballroom B

10:30AM AT.00001 Optimization of Kinematics of a Flapping Wing Mechanism\(^1\). SCOTT THOMSON, CHRISTOPHER MATTSON, MARK COLTON, MIKE TREE, Brigham Young University — Flapping flight offers several potential advantages over conventional fixed wing flight, such as agility and maneuverability in confined spaces, potentially decreased noise and detectability, and hovering capability. In this presentation, a water tunnel-based flapping wing apparatus is introduced that allows for arbitrary wing trajectories in three rotational degrees of freedom and simultaneous measurements of lift and thrust production. An optimal flapping trajectory for takeoff is found using hardware-in-the-loop optimization methodology. Wing motion derived from high-speed imaging of a ladybug during takeoff is used as a first iteration of the hardware-in-the-loop optimization. Using real-time force measurements and a gradient-based optimization approach, the algorithm searches for the optimal trajectory for a variety of parameters such as lift or efficiency. Hardware performance is assessed. Results from the optimization routine, including the final flapping trajectory are reported for both rigid and compliant wings.

\(^1\)Research funding from the Air Force Office of Scientific Research is gratefully acknowledged.

10:43AM HT.00002 Aerodynamic tricks for pitching oscillation and visual stabilization in a hovering bird. JIAN-YUAN SU\(^1\), SHANG-CHIEH TING, JING-TANG YANG, Department of Mechanical Engineering, National Taiwan University, Taipei 10617, Taiwan, BEAM LAB TEAM — We experimentally investigate how small birds attain a stabilized vision and body posture during hovering. Wing-beats of finches and passerines executing asymmetrical hovering provide lift merely during the downstroke. The downstroke lift is significantly greater than the bird weight, thereby causing a pitch-up swing of the bird body. A hovering bird skillfully and unceasingly tunes the position and orientation of lift force to stabilize its vision, so that the eye displacement is approximately one-tenth less than the tail, causing an illusion that the bird body is rotating about the eye. The hovering birds also spread and fold periodically their tail with an evident phase relationship with respect to the beating wings. We found that hovering birds use their tail to intercept the strong downward air-flow induced by the downstroking wings, and sophisticatedly spread their tail upon the arrival of the downward air-flow, rendering a pitch-up moment that effectively counteracts the pitch-down body rotation. Hence during hovering the bird essentially undergoes a dynamically-stable pitching oscillation, and concurrently attains a stabilized vision.

\(^1\)+886-2-3366-2722

10:56AM HT.00003 Optimal frequency for flow energy harvesting using flapping foils and its relation with wake instability. QIANG ZHU, UC San Diego — Inspired by the correlation between the propulsion efficiency of a flapping foil and stability of the wake behind it (which leads to the optimal Strouhal number for propulsion), we numerically simulated a foil in energy harvesting mode, and investigated the relation between wake stability and the energy harvesting efficiency (defined as the portion of incoming flow energy extracted by the system). The base flow is computed using a Navier-Stokes algorithm and the flow stability analysis is performed numerically via the Orr-Sommerfield equation. The wake is found to be convectively unstable and the frequency of the most (spatially) unstable mode \( f_w \) is determined. The optimal efficiency occurs when \( f_w \) is close to \( f \) (the oscillation frequency of the foil), which is achieved when \( f \) is close to 0.15 (hereby \( f \) is normalized by the chord length and the speed of incoming flow). In addition, for this “foil-wake resonance” to happen there must be significant leading edge separation associated with large effective angles of attack.

11:09AM HT.00004 The advantage of wing-wing interaction in unsteady motion. TUYEN QUANG LE, Konkuk University, DOYOUNG BYUN, SOO HYUNG PARK, Konkuk University, JIN HWAN KO, Seoul National University, HOON CHOEIL PARK, Konkuk University — The role of elytra in aerodynamic performance of flapping flight has been numerically investigated for beetle flight. In a case of hovering flight, the relatively small vertical or horizontal forces were generated by the elytra and no significant contribution to aerodynamic force from elytra and hindwing interaction of Coleopteran insect. On the other hand, the flapping elytra may increase the total force around 20% on both wings by the wing-wing interaction such as flow blocking and flow acceleration between the wings in forward flight. The flow blocking and acceleration strongly depends on phase angle, gap between wings. Additionally, the optimal condition for thrust force generation and aerodynamic efficiency was found from parameter study of in- and out-phase angles combined with gap between two airfoils.

11:22AM AT.00005 Lift generation by a two-dimensional symmetric flapping wing. TAKAJI IN-AMURO, KEIGO OTA, KOSUKE SUZUKI, Department of Aeronautics and Astronautics, Kyoto University — Two-dimensional symmetric flapping wing is investigated by an immersed boundary-lattice Boltzmann method. In the method we can treat the moving boundary problem efficiently on the Cartesian grid. First, we investigate the effect of the Reynolds number on flows around symmetric flapping wings under no-gravity field and find that at high Reynolds numbers symmetric vortices are appeared and the time-averaged lift force is induced on the wings, while at low Reynolds numbers only symmetric vortices are appeared around the wings and no lift force is induced. Also, the effect of the initial position of the wings on the lift force is investigated. Secondly, we carry out free flight simulations under gravity field for various Reynolds and Froude numbers and find the region where upward flights are possible.
A low-order inviscid point vortex model is used to simulate the pitching and perching motion of a thin flat plate at low Reynolds number. These motions induce coherent vortex shedding at the leading edge, which has a profound influence on the generated force. The low-order method is based on the inviscid Brown-Michael point vortex model, which accounts for the unsteady aerodynamics by tracking a small number of vortices with time-varying strengths. For the pitching motion, the results from the low-order model are compared with high fidelity simulations under different pitching rate and axis position, and this comparison shows a good qualitative agreement.

The perching motion is characterized by larger rotations and an unsteady translation. The low-order model results are compared with previous experiments conducted in a water tunnel, and a good qualitative agreement is achieved. To investigate the mechanism of force generation, the force obtained from the model is decomposed into inertial reaction and circulatory components, and their relative contributions are inspected.

We acknowledge the French Research Agency for support through Project No. ANR-08-BLAN-0009.

1Department of Mechanical Engineering National Taiwan University

10:30AM HU.00001 Perturbed Partial Cavity Drag Reduction at High Reynolds Numbers

11:35AM HT.00006 Low-order Modeling of Bio-inspired Pitching and Perching at Low Reynolds Numbers

12:01PM HT.00008 Elasticity Estimation of Thin Flap Using Optical PIV Velocity Fields

12:14PM HT.00009 Analysis of the Flapping Dynamics of a Slender Within a Soap-Film Flow Tunnel

12:27PM HT.00010 Wing compliance in self-propelled flapping flyers

Monday, November 22, 2010 10:30AM - 12:40PM –
Session HU Multiphase Flows V

Hyatt Regency Long Beach Regency A
10:43AM HU.00002 Stability Analysis of Superhydrophobic Friction Reduction Polymeric Microchannels, TAE JIN KIM, CARLOS HIDROVO, The University of Texas at Austin — Superhydrophobic surfaces are surfaces where fluid contact angle is larger than 150°. Superhydrophobic states which allow water droplets to fall off at low sliding angles are termed as Cassie state. It is widely known that drag/friction reduction is closely related to liquid under Cassie state, and studies have been widely performed to achieve such effects. Our research goal is to develop superhydrophobic microfluidic channels with trenches on the side walls and observe the stability of the air pockets formed within these trenches. We have prepared PDMS(poly-dimethylsiloxane) substrates with different trench aspect ratio of 1:1, 1:2, 1:500 and 1:3000. As the aspect ratio of the trench decreases, the pressure in the air pockets tends to resist wetting. However, once penetration of the water into the air pocket occurred, the shallow trenches were wetted in a rapid fashion while the deep trenches were wetted at a slower rate. A compression model of the air pockets as a function of pressure difference and volume change of the air pockets was also developed. In the theoretical model, the air in the pockets is assumed to be an ideal gas. This model was compared and validated against the experimental results.

10:56AM HU.00003 Heating Effects In Very Rough Polymeric Microchannels, PHILLIP GLASS, The University of Texas at Austin, ARNAV CHHABRA, RAVITEJ KANAPURAM, University of Texas at Austin, TAE JIN KIM, CARLOS HIDROVO, The University of Texas at Austin — Slip in internal flows is known to reduce friction and thus reduce the required pumping power. One method to achieve slip is by ‘roughening’ the surface to induce Cassie state. The Cassie state is a phenomenon in which a liquid rests on top of a rough surface with a gas layer formed underneath. Our research goal is to develop a highly rough microfluidic channel and study the heating effects on the air pockets trapped between the roughness elements. We have prepared a PDMS(poly-dimethylsiloxane) microfluidic channel with trenches on the side walls. The channel dimension is 100um x 110um (width x height), and the dimensions of each trench are 30um x 60um x 110um (width x length x height). As the heat flux into the microfluidic channel increases the air trapped on the trenches expanded increasing the volume of the void. In order to prevent the expanding air from invading the liquid flow layer the pressure drop was increased. Therefore by heating the channel the wetting of air pockets can be prevented even under higher pressure drops, thus maintaining the two phase flow and significantly reducing the friction coefficients.

11:09AM HU.00004 Micro-PIV Measurements near a Moving Contact Line, JEREMIAH ZIMMERMAN, MARK WEISLOGEL, DEREK TRETHEWAY, Portland State University — The displacement of one fluid by an immiscible second fluid (i.e. dynamic wetting), governs many natural and technological processes. Despite extensive studies, understanding and modeling the displacement process remains one of the outstanding problems in fluid mechanics. In this work, we explore the physics of the moving contact line (the idealized line of intersection between two fluids and a solid) by measuring velocities near the moving contact line with micron resolution particle image velocimetry. The measured flow is generated by dynamic wetting in a glass microchannel. The microchannel is mounted on an automated microscope stage with precise velocity control allowing for the static placement of the contact line within the field of view. Full-field velocity measurements near the contact line were made in water/glycerol and fructose/glucose/water solutions. Preliminary results appear to show remarkable similarity to controversial theoretical predictions.

11:22AM HU.00005 Film Deposition in the Presence of a Moving Contact Line1, ALEXANDRU HERESCU, JEFFREY S. ALLEN, Michigan Tech University — Film deposition experiments are performed in circular glass capillaries of 500 μm diameter. Two surface wettabilities are considered, contact angle θ = 30° for water on glass and θ = 105° when a hydrophobic coating is applied. It was observed that the liquid film deposited as the meniscus translates with a velocity U presents a ridge which also moves in the direction of the flow. The ridge is bound by a contact line moving at a velocity UCL, as well as a front of velocity UF, and it translates over the deposited stagnant film. The behavior of the ridge presents striking dissimilarities when the wettability is changed. Both UCL and UF are approximately twice as large for the non-wetting case at the same capillary number Ca. Classical film deposition theory does not account for the existence of a contact line and it assumes perfect wetting. In contrast, the contact line dynamics fundamentally alter the deposition physics by causing the film to be non-stagnant. As a consequence the non-wetting film is significantly thicker than the Bretherton prediction. Taylor bubbles also form due to the growth of the ridge and are differentiated by wettability, being much shorter and presenting a thicker film in the non-wetting case. The dynamics of the contact line is studied experimentally and a criterion is proposed to explain the occurrence of a shock in the non-wetting film.

1We would like to express our thanks to the NSF for the support of this work.

11:35AM HU.00006 Optical characterization of fibers suspensions in turbulent pipe flow, STELLA DEARING, ALFREDO SOLDATI, Dept. Energy Technologies, University of Udine — Suspensions of elongated rigid fibers in turbulent flows are commonly encountered in applications of engineering interest, and may exhibit complicated rheological properties depending on the spatial distribution and orientation of the fibers. Despite the practical importance of fiber suspensions there is insufficient experimental data to validate numerical simulations and provide benchmarks. This paper presents an image analysis algorithm used to calculate orientation and distribution of fibers suspended in turbulent pipe flow. The algorithm is validated using artificial images. These images represent three-dimensional randomly orientated ellipsoids illuminated by a laser sheet and projected onto a two-dimensional plane. The error magnitude on the orientation distribution and number density is found by means of Monte Carlo simulations. Experiments are carried out considering small control volume near the pipewall. Results indicate that fibers exhibit preferred spatial orientation close to the pipewall and more randomized orientation close to the centerline, in qualitative agreement with the available numerical simulations.

11:48AM HU.00007 On the periodic motion of a disk falling freely in a tube, NICOLAS BROUSSE, PATRICIA ERN, Institut de Mecanique des Fluides de Toulouse, France — Freely falling or rising particles in an unconfined low-viscosity fluid otherwise at rest are known to exhibit oscillatory motions, such as helicoidal or zigzag paths. In this work, we characterized experimentally the oscillatory motions of disks falling in a tube at Reynolds numbers 60 < Re < 250, covering both rectilinear and periodic motions. The fall of the bodies (of density close to that of the fluid) was followed by two travelling cameras to determine the body’s translation and rotation characteristics. We focused on the effect of the confinement factor (ratio of the diameter of the body, d, to that of the tube, D). The study was carried on for different body’s aspect ratio (ratio of its diameter d to thickness h, taken as 3, 6 and 10), since this parameter is known to strongly influence the characteristics of the oscillatory motions observed in the unconfined situation.

12:01PM HU.00008 Stretching of flexible molecules inside fluid threads, PAULO E. ARRATIA, GABRIEL JUAREZ, University of Texas at Austin — The evolution of viscoelastic fluid threads undergoing capillary breakup is complex and depends on the delicate balance between capillary, viscous, and elastic stresses which result in behavior that is markedly different from Newtonian fluids such as the “beads-on-a-string” phenomenon. Here, we aim to understand the thinning of a fluid thread and the drop breakup process of polymeric fluids in a simple microfluidic device by direct visualization of fluorescent DNA molecules. Molecules are observed to transition from a coiled state to an almost fully stretched state when experiencing extensional flow within the filament. The stretching of flexible molecules under applied viscous stress is characterized by a simple worm-chain model.
11:30AM HV.00003 High-Order Quasi-Steady State Assumption for Chemistry Reduction
ASHRAF IBRAHIM, SHARATH GRIMAJI, Texas A&M University — The quasi-steady state assumption (QSSA) is one of the most physically compelling concepts used for reducing large chemical kinetic mechanisms. However, QSSA is not helpful in chemistry regimes where experiential knowledge of the kinetic set behavior is lacking. This has lead to the development of more advanced kinetics reduction schemes which, while mathematically precise, are physically less insightful than QSSA. In this work, we develop a higher-order QSSA (HO-QSSA) formulation which is mathematically precise while preserving the physical clarity of the original QSSA. The talk will present examples of chemistry reduction and demonstrate the connection between HO-QSSA and other currently used reduction schemes such as Intrinsic Low Dimensional Manifold (ILDM).

12:27PM HU.00010 Improved flying hot-film anemometry in liquid-gas flows
SANTOS MENDEZ-DIAZ, ROBERTO ZENIT, JUAN HERNANDEZ-CORDERO, Instituto de Investigaciones en Materiales, UNAM, REOLOGIA TEAM — A modified hot-film anemometry technique was used to measure liquid velocity fluctuations resulting from bubble agitation in a liquid-gas flow. The first modification aims to remedy the main drawback in hot-film anemometry measurements in liquid-gas flow: bubble-probe interaction. To improve bubble detection, optical fibers were installed in close proximity to the anemometer sensing element; in this way, the collisions of bubbles with the probe can be detected and removed from the signal. The second modification resolves the poor performance of the probe at small mean liquid velocity. The sensing element is moved at a known rate; subsequently, this translation velocity is removed from the signal leaving only the fluctuating velocity of the liquid. Furthermore, an analysis of the effect of the signal processing parameters, such as detection and signal length threshold, is conducted. The flow conditions at which this technique was tested covered void fractions up to 6% in nearly monodispersed bubbly flows. The results obtained show good agreement with reported data by other authors in both, variance and spectral density of the liquid velocity. This technique can be used to measure pseudoturbulence in on bubbly flows.

Monday, November 22, 2010 10:30AM - 12:40PM
Session HV Reacting Flows I Hyatt Regency Long Beach Regency B

12:14PM HV.00005 An ODT-Based Flame-Embedding Approach for Turbulent Non-Premixed Combustion
SUMIT SEDHAI, TAREK ECHEKKI, North Carolina State University — A multiscale formulation is implemented to capture finite-rate chemistry in turbulent non-premixed flames. The formulation is based on a coupling of a large-eddy simulation (LES) solution for the transport of the filtered mixture fraction field and one-dimensional solutions embedded on the flame brush, which are tracked with the stoichiometric value of the filtered mixture fraction. The one-dimensional solutions are part of many industrial processes. The oil drops size and dispersion stability are determined by the impeller geometry, stirring velocity and the physicochemical properties of the mixture. A critical parameter is the total interfacial area which is increased as the drop size is decreased. The mechanism that disperses the oil and generates the drops has not been completely explained. In the present work, castor oil (1% v/v, viscosity 500mPa) and water are stirred with a Scaba impeller in a flat bottom cylindrical tank. The process was recorded with high-speed video and the Reynolds number was fixed to 24,000. Before the stirring, the oil is added at the air water interface. At the beginning of the stirring, the oil is suctioned at the impeller shaft and incorporated into the flow ejected by the impeller. In this region, the flow is turbulent and exhibits velocity gradients that elongate the oil phase. Viscous thin filaments are generated and expelled from the impeller. Thereafter, the filaments are elongated and break to form drops. This process is repeated in all the oil phase and drops are incorporated into the dispersion. Two main zones can be identified in the tank: the impeller discharge characterized by high turbulence and the rest of the flow where low velocity gradients appear. In this region surface forces dominate the inertial ones, and drops became spherical.

10:43AM HV.00002 Dynamics in reactive bubbly flow
PAVITHRA SUNDARARAJAN, DONALD KOCH, ABRAHAM STROOCK, Cornell University — Multiphase flow in microfluidic channels encompasses a rich collection of phenomena of widespread interest in both fundamental and technological context. While studies on non reactive multiphase flow focus on the dynamics of bubble breakup, coalescence and stability, a reactive multiphase flow opens up a broader spectrum of dynamics, like nucleation, growth and detachment of bubbles as well as the secondary mixing in the slugs during these processes. Our interest lies in the flow in an electrochemical microfluidic fuel cell with liquid reactants reacting at catalyst walls producing gaseous products which choke the fuel cell efficiency due to uncontrolled bubbly flow. This challenge is an opportunity in itself provided the multiphase flow dynamics can be characterized to achieve a stable Taylor regime. Taylor regime allows for promisingly high efficiencies due to improved mass transfer of reactants to the concentration boundary layer of the electrodes achieved by the secondary flow in the liquid phase present between bubbles. Here, I will experimentally explore the different regimes of reactive bubbly flow in a microchannel. The phase diagram of the reactive multiphase flows would be used to identify the stable regime for efficient fuel cell operation. Further, I will study the mass transfer in the presence of multiphase flow to regimes of enhanced mass transfer, and compare it with numerical models.

10:56AM HV.00003 An ODT-Based Multiscale Radiative Transport Model in Participating (absorbing-emitting) gray media
YAJUVENDRA SHEKHAWAT, TAREK ECHEKKI, North Carolina State University — A multiscale formulation for thermal radiation transport in participating (absorbing-emitting) gray media is developed. The model is based on a grid topology using the one-dimensional turbulence (ODT) model framework and the photon Monte-Carlo (PMC) method for radiative transport. The formulation is implemented within the context of large-eddy simulation (LES). The ODT solution for the evolution of temperature field are based on, (a) a deterministic implementation for diffusion, advection and reaction, and (b) a stochastic implementation for LES subgrid scale advective transport. Specific rules for ray tracing and the modeling of emission and absorption processes are designed to capture turbulence-radiation interactions and account for subgrid scale contribution due to residual temperature fluctuations. The model is implemented for turbulent premixed flame problem and compared with DNS. The results yield excellent agreement between the LES-ODT model for radiative transport and DNS predictions.

11:09AM HV.00004 Simulation of Compressible Reacting Flow using the Parallel Wavelet Adaptive Multiresolution Representation
ZACHARY ZIKOSKI, SAMUEL PAOLUCCI, JOSEPH M. POWERS, University of Notre Dame — The Wavelet Adaptive Multiresolution Representation (WAMR) provides spatial adaptivity which automatically supplies local grid resolution based on the demands of the solution. The WAMR method allows for a wide range of spatial scales to be captured while minimizing the number of degrees of freedom needed in the solution. Recently, the WAMR algorithm has been adapted for use on massively parallel computer architectures using an MPI-based domain decomposition approach. Results for parallel performance on test problems will be presented. Additionally, results from the application of the parallel WAMR algorithm to multidimensional compressible, reacting flows will be shown. Calculations include propagation of a detonation in a square channel and a combustible mixing layer using detailed chemistry models.

11:22AM HV.00005 An ODT-Based Flame-Embedding Approach for Turbulent Non-Premixed Combustion
SUMIT SEDHAI, TAREK ECHEKKI, North Carolina State University — A multiscale formulation is implemented to capture finite-rate chemistry in turbulent non-premixed flames. The formulation is based on a coupling of a large-eddy simulation (LES) solution for the transport of the filtered mixture fraction field and one-dimensional solutions embedded on the flame brush, which are tracked with the stoichiometric value of the filtered mixture fraction. The one-dimensional solutions are based on the one-dimensional turbulence (ODT) model. In the ODT solutions for the evolution of the reactive scalars, diffusion and reaction processes are implemented deterministically while LES subgrid scale advective transport have stochastic implementation. The ODT domains are allowed to be advected with the flow such that they remain attached to the flame brush. The formulation enables the implementation of effects that capture the fluctuations of reactive scalars within the LES grid, including finite-rate and non-equilibrium chemistry effects and radiation-turbulence interactions.
11:35AM HV.00006 A dynamic approach for nonequilibrium modeling of subfilter scalar dissipation rate in combustion LES . COLLEEN M. KAUL, VENKAT RAMAN, The University of Texas at Austin — The filtered scalar dissipation rate is a fundamental parameter in combustion LES, appearing as an input parameter in all combustion models. Since subfilter dissipation is a small scale quantity, conventional dynamic modeling approaches are not valid. Typically, this quantity is obtained by assuming that the production of scalar variance at filtered scales is exactly balanced by dissipation, leading to an algebraic relation for the dissipation rate. However, this local equilibrium assumption is highly restrictive since it neglects spatial transport. Here, we propose a new modeling approach that overcomes this limitation. This nonequilibrium model uses a dynamic approach along with the scalar variance transport equation to determine the dissipation rate. A priori studies using DNS are used to evaluate the accuracy of the method. In addition, a novel a posteriori method is used to assess model performance in LES calculations.

11:48AM HV.00007 Quantitative Visualization of transverse annular jets , BRIAN VENTURA, California Institute of Technology, KLULAI CHOW-YEE, UC Berkeley, JASON DAMAZO, PHILIPP BOETTCHER, JOSEPH SHEPHERD, California Institute of Technology, IOANNIS MIKELLIDES, DAVID VAUGHAN, NASA-JPL — Transverse injection of fluid into an annular jet is a mechanism resulting in good mixing and is therefore utilized in engineering applications such as pintle rocket engines. Vigorous mixing occurs between the two jets. However, much of what we know about the flow behavior of such devices has been learned empirically with very limited studies exploring the fluid dynamics. The geometry under investigation is an asymmetric radial jet of variable width impinging on a fixed annular jet. The main capability of the current facility is to reproduce start-up and quasi-steady flow conditions through the use of a fast acting valve which opens a pressurized air reservoir. The flow is then observed using laser interferometry giving quantitative measurements of the density fields that are compared with computations. The main fields under investigation were the reservoir pressure and the area ratio between the axial and radial jet. Modeling of the fluid mechanics was performed at NASA-JPL.

12:01PM HV.00008 Ex vivo Characterization of Blast Wave Impact and Spinal Cord Tissue Deformation , JUN CHEN, JIAN GAO, SEAN CONNELL, RIYI SHI, Purdue University — Primary blast injury on central nervous system is responsible for many of the war related casualties and mortalities. An ex vivo model system is developed to introduce a blast wave, generated from a shock tube, directly to spinal cord tissue sample. A high-speed shadowgraph system is utilized to visualize the development of the blast wave and its interaction with tissue sample. Surface deformation of the tissue sample is also measured for the analysis of internal stress and possible injury occurred within the tissue sample. Understanding the temporal development of the blast-tissue interaction provides valuable input for modeling blast-induced neurotrauma. Tracking the sample surface deformation as a function of time provides realistic boundary conditions for numerical simulation of injury process.

12:14PM HV.00009 Experimental study on intensity of change in viscosity by a chemical reaction on a liquid flow . SHOHEI IWATA, YUICHIRO NAGATSU, YOSHIHITO KATO, YUTAKA TADA, Nagoya Institute of Technology, Japan — We have recently investigated a liquid flow involving viscosity change by chemical reactions. The liquid flow involves the displacement of the more viscous liquid by the less viscous one in a Hele-Shaw cell. So far, we reported the effects of Damkohler number (Da), which is defined as the ratio between a characteristic time of fluid motion and that of chemical reaction, on the flow. In reacting liquid flows involving viscosity changes, an intensity of change in viscosity, as well as Da, is supposed to be important. In the present study, we have experimentally investigated the effect of the intensity of decrease in viscosity by the chemical reaction on the liquid Hele-Shaw flow for a condition of infinite Da. We have found a threshold value of the intensity, beyond which the flow is dramatically changed by the reaction.

12:27PM HV.00010 Buoyancy-driven instabilities of acid-base fronts , CHRISTOPHE ALMARCHA, IRPHE - UMR 6594 - CNRS - Université de Provence - Aix-Marseille Université, France, YASMINA R’HONI, PHILIP M.J. TREVELYAN, ANNE DE WIT, NLPC, Université Libre de Bruxelles, Belgique — Chemical reactions can produce buoyancy-driven motions in solutions by changing the local density in the gravity field. Starting from a stratification of one given miscible reactant solution on top of another miscible one, convective destabilization influenced by the reaction can emerge as a combination of several non-reactive hydrodynamic instabilities, including Rayleigh-Taylor and double diffusive instabilities. In the specific case of reactions between a strong acid and a strong base, we show that all the possible scenarios reduce to the composition of only two asymptotic situations because products generated at the reactive zone are always less dense and slower diffusing than the reactants. Experiments in a vertically oriented Hele-Shaw cell confirm the theoretical predictions and can be quantitatively compared to numerical simulations of a nonlinear reaction-diffusion-convection model.

Monday, November 22, 2010 10:30AM - 12:40PM – Session HW Surface Tension II Hyatt Regency Long Beach Regency C

10:30AM HW.00001 Elastocapillary snapping , ARNAUD ANTKOWIAK, CNRS & UPMC Univ Paris 06, Institut Jean Le Rond d’Alembert, Paris, France, AURELIE FARGETTE, Ecole Normale Superieure, Departement de Physique, Paris, France, SEBASTIEN NEUKIRCH, CNRS & UPMC Univ Paris 06, Institut Jean Le Rond d’Alembert, Paris, France — An elastica buckled in the form of an arch is subjected to a transverse force. Above a critical load value, the buckling mode is switched and the elastica takes the form of a reversed arch. This is the well-known snap-through phenomenon which has been extensively studied in solid mechanics. Here, we revisit this phenomenon and show that capillary forces may promote snapping of a buckled polymer strip. We report detailed experiments of this new paradigm for elastico-capillary interactions, and the obtained results are in close agreement with a simple elastic stability theory.

10:43AM HW.00002 Capillary bond between rod-like microparticles at interfaces , LORENZO BOTTO, KATHLEEN J. STEBE, University of Pennsylvania, Chemical and Biomolecular Eng. Dept. — Elongated microparticles at a fluid interface create interface distortions, which influence the surrounding fluid. We develop a numerical model for the transport of microparticles in the fluid phase while microparticles have been studied -ellipsoids, which are predicted to assemble side-to-side, and cylinders, which chain end-to-end. The differences can be attributed to near field interactions, which we term the capillary bond. We simulate the capillary bond between two elongated particles as a function of inter-particle separation, relative orientation, and particle shape. The particle is represented as a super-ellipsoid, a parameterization which allows the study of a broad class of shapes, from ellipsoids to cylinders with rounded or sharp corners, upon varying a single parameter. The geometric details of the particles have a dramatic effect on the dependence of the capillary bond on the configuration, a finding with strong consequences for micro-structures formed by these particles.

10:56AM HW.00003 Using Convective Flow to Reach the Kinetic Limit of Surfactant Transport to a Stationary Microbubble , NICOLAS ALVAREZ, Chemical Engineering Department, Carnegie Mellon University, DOUG VOGUS, Chemical Engineering Department, Bucknell University, LYNN WALKER, Chemical Engineering Department, Carnegie Mellon University, SHELLEY ANNA, Chemical Engineering Department, Mechanical Engineering Department, Carnegie Mellon University — Surfactant transport is characterized near microscale interfaces in the presence of flow. We infer transport mechanisms by simultaneously measuring the radius of curvature and the pressure jump across the interface of a micron size bubble. The surfactant is dissolved in the liquid surrounding the bubble. Flow is introduced to minimize concentration gradients caused by diffusion. The dynamic surface tension is monitored at initially clean interfaces for different flow rates. The kinetic limit is achieved as the flow rate increases.

The observed dynamics are interpreted in the context of a scaling analysis and a one dimensional convective transport model. This device and theory will be instrumental in measuring and modeling kinetic exchange dynamics at fluid-fluid interfaces for more complex surface-active species, including mixed surfactants, polyelectrolytes, and biomolecules.
11:09 AM HW.00004 Wicking flow through microfluidic channels, HADI MEHRABIAN, PENG GAO, JAMES J. FENG, University of British Columbia — Diffuse-interface models can be used to simulate contact line motion on solid substrates by regularizing the singularity by diffusion. Using the Cahn-Hilliard model and a finite-element algorithm, we have computed wicking flows in microfluidic channels of three types of geometries. The first type features axisymmetric tubes with contractions and expansions of the cross section. Both drainage and imbibition dynamics are studied, and we define critical conditions for the contact line to negotiate sharp corners on the wall. The second type consists of bifurcations in micro-channels where the competition between capillary pressure in the branches and viscous loss in the feeding tube produces different flow patterns. Finally, we examine tortuous channels in Z and U-shaped turns, where the effect of streamline on the flow rate is analyzed as a prototype for tortuosity in porous medium.

11:22 AM HW.00005 Migration of droplets driven by thermocapillary stress, JUAN M. GOMBA, IFAS, UNCPBA, Argentina, GEORGE M. HOMSY, Dept. Math. University of British Columbia, Canada — We study the effect of wettability on the flow of droplets driven by thermocapillary effects. An equation for the thickness profile of the droplet is derived by employing lubrication approximations. The model includes the effect of the contact angle introduced through a disjoining-conjoining pressure term. For complete wetting or low contact angles, the droplet spreads into a long film profile with a capillary ridge near the leading edge, a behaviour that resembles the experiments on Marangoni films reported by Ludviksson & Lightfoot (1971). A self similar solution for the profile of the film and an expression for the non constant velocity of the leading edge are presented. For high contact angles, the droplet moves with a constant velocity as a single entity. Here, the effect of the disjoining pressure is strong enough to keep the droplets almost undistorted from its static shape. This regime is the usual one reported in experiments on thermocapillary migration of droplets. An expression for the velocity is derived. For intermediate values of the contact angle the Marangoni stress and the disjoining-conjoining pressure compete and, accordingly, the behaviour is transient and complex. The occurrence of these three regimes and their dependence on various parameters is analyzed.

11:35 AM HW.00006 On the structure of Marangoni-driven singularities, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — This work presents an analytical study of the structure of steady Marangoni-driven singularities. While the results are applicable to a wide class of phenomena, the analysis is performed on the example of tip-streaming, which is driven by chemical-reaction producing a surfactant at the interface of a two-phase system. Due to the conical symmetry of the problem, there exist self-similar solutions of the Stokes equations, which are singular at the tip and thus provide no information on the thread structure which is responsible for tip-streaming. This cone-tip singularity is resolved with the help of asymptotic matching of the self-similar and thread solutions using thin layer (slender jet) approximation, which gives explicit asymptotic formulas for the scaling of the thread radius and thus of the emitted droplets as a function of physical parameters.

11:48 AM HW.00007 On the true nature of chemical reaction-driven tip-streaming, HANS C. MAYER, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — In the course of recreating the experiments of J. Fernandez and G.M. Homsy [Phys. Fluids 16, 2548 (2004)] on chemical reaction-driven tip-streaming in a pendant drop, we identified the true source of this ‘amazing’ phenomenon and thus provided physical interpretation to the unexplained observations in the aforementioned work. This finding added a new parameter dimension to the problem and led to its more complete parametric study. In particular, we studied the effect of the key physical parameters (chemical reaction rate, surfactant properties, etc.) on the emission frequency and size of drops, emitted in both the oscillating and steady tip-streaming modes. Altogether, the conducted study made the experiments more controllable and understanding of the phenomena less “elusive.”

12:01 PM HW.00008 A VOF-based method for the simulation of thermocapillary flow, CHEN MA, DIETER BOTHE, Technical University Darmstadt — This contribution concerns 3D direct numerical simulation of surface tension-driven two-phase flow with free deformable interface. The two-phase Navier-Stokes equations together with the energy balance in temperature form for incompressible, immiscible fluids are solved. We employ an extended VOF (volume of fluid) method, where the interface is kept sharp using the PLIC-method (piecewise linear interface construction). The surface tension, modeled as a body force via the interface delta-function, is assumed to be linearly dependent on temperature. The surface temperature gradient calculation is based on carefully computed interface temperatures. Numerical results on thermocapillary migration of droplets are obtained for a wide range of Marangoni numbers. Both the terminal and initial stage of the migration are studied and very good agreement with theoretical and experimental results is achieved. In addition, simulation of the Bénard-Marangoni instability in square containers with small aspect ratio and high-Prandtl-number fluids is discussed concerning the development and numbers of convection cells in relation to the aspect ratio.

12:14 PM HW.00009 Capillary interactions between elongated microparticles: A Pair Potential, KATHLEEN STEBE, ERIC LEWANDOWSKI, LORENZO BOTTO, UPenn — Particles at fluid interfaces create distortions. When distortions from neighboring particles overlap, the area, and hence the energy, decreases if the particles migrate toward each other. Elongated particles orient as they approach, and have preferred orientations upon assembly. For elongated microparticles like ellipsoids or cylinders, the interface distortion resembles an elliptical quadrupole a few radii away from the surface particle. We present an anisotropic pair potential based on elliptical quadrupoles. This potential predicts an attractive force and a torque, which depend strongly on aspect ratio, in keeping with experiment on cylinders at interfaces. Particle trajectories and angular orientations recorded by video microscopy for cylinders agree with the predicted potential. In particular, the analysis predicts the rate of rotation, a feature lacking in prior analyses. Open issues associated with near field effects are briefly discussed.

12:27 PM HW.00010 Colloidal dynamics near an interface, MADHAV MANI, Harvard University & KITP, UCSB, VINOTHAN MANOHARAN, MICHAEL BRENNER, DAVID KAZ, RYAN MCGORTY, Harvard University — Although the equilibrium state of a colloidal particle at an interface is well understood, the dynamics associated with the approach to equilibrium is not. Recent high-resolution experiments have shown that the dynamics are richer than expected. This part of the study focuses on the evolution of the system after the initiation of a contact-line. We model the dynamics associated with three different regimes: the center of mass (c.o.m.) motion, the location of the contact-line and the dynamic contact-angle. Following Nikolov et al. (Journal of Colloid and Interface Science - 112,1,1986), we derive the statements of force balance by taking variations of an energy functional. Appealing to a balance of power we are able to derive the dynamical laws. Associated with the degrees of motion are three modes of dissipation corresponding to a moving c.o.m., a moving contact-line and an evolving contact angle. We derive an asymptotically valid model for the system, which we integrate numerically and compare to experiments.

Monday, November 22, 2010 10:30AM - 12:40PM
Session HX Compressible Flows II
Hyatt Regency Long Beach Regency D

10:30 AM HX.00001 Low-order stochastic model for the low-frequency shock motions in shock/boundary-layer interactions, EMILE TOUBER, Imperial College London, NEIL SANDHAM, University of Southampton — The need for better understanding of the low-frequency unsteadiness observed in shock wave/turbulent boundary layer interactions has been driving research in this area for several decades. Starting from an exact form of the momentum integral equation and guided by large-eddy-simulation data, a stochastic ordinary differential equation for the reflected-shock foot low-frequency motions is derived. The frequency of the most energetic fluctuations is shown to be a robust feature over a wide range of input parameters in agreement with experimental observations. Under some plausibility assumptions, the coupling between the shock and the boundary layer is mathematically equivalent to a first-order low-pass filter. Therefore, it is argued that some observed low-frequency unsteadiness is not necessarily a property of the forcing, either from upstream or downstream of the shock, but simply an intrinsic property of the coupled dynamical system.
10:43AM HX.00002 Shock Wave–Boundary Layer Interaction in Reflecting Detonations

J. DAMAZO, J. ZEIGLER, J. KARNESKY, J.E. SHEPHERD, California Institute of Technology — The interaction of a reflecting shock wave with the boundary layer induced by the incident shock wave results in a unique flow field that has been examined in shock tubes. Our recent experiments studying reflecting detonations examine an incident detonation impinging on a normal, planar wall to create a reflected shock wave. We have observed that the pressure records taken near the location of reflection show that the measured speed of the reflected shock wave is inconsistent with the measured wall pressures. We present new experimental results of high-speed video taken of the reflecting detonation and high-resolved two-dimensional numerical simulations of compressible viscous flow. These results show that the interaction of the reflected shock wave with the boundary layer can result in a three-dimensional shock front structure with an oblique front in the boundary layer similar to that observed in non-reacting shock tubes.

10:56AM HX.00003 Techniques for PIV-CFD Comparison in a Shock-boundary Layer Interaction

DAVID HELMER, JOHN EATON, Stanford University — The outputs of a PIV experiment and a simulation of the same system are fundamentally different. PIV includes spatial averaging effects, particle travel effects, sampling effects, and multiple potential biases. In addition, there are many uncertainties, some of which are poorly understood or documented. These issues make direct comparisons between the two fields inappropriate in certain cases. To address this, a hierarchy of techniques for performing meaningful comparisons between simulations and experiments is described. The simplest transformation incorporates only spatial averaging. At the highest level of fidelity, a method is presented that links high-fidelity results directly from a PIV-CFD comparison in this system are discussed, along with the necessary steps required to transform a CFD simulation of this system into a field that can be directly compared to the PIV.

11:04AM HX.00004 Upstream Boundary Condition Sensitivity of the Shock-Boundary Layer Interaction

LAURA CAMPO, DAVID HELMER, TONKID CHANTRASMI, GIANLUCA IACCARINO, JOHN EATON, Stanford University — A low aspect ratio Mach 2.1 wind tunnel with a wall-mounted compression wedge is being used to validate uncertainty quantification techniques for CFD. The tunnel is operated continuously, with a mass flow rate of approximately 0.6 kg/s. The incoming pressure, temperature, and mass flow rate are monitored, and the variation in these boundary conditions is documented to provide bounds for the fluctuations applied in the CFD. The compression wedge generates an oblique shock, resulting in flow separation at the base of the wedge. High-resolution PIV measurements are taken throughout the field, with a focus on the shock-boundary layer interaction at the base of the compression wedge and on the location of shock impingement on the opposite wall. The boundary layer is perturbed in a Mach 2.1 flow field by the compression wedge, and the interaction is studied using high-resolution PIV measurements. The perturbed velocity field is measured at the location where the oblique shock from the compression wedge impinges upon the opposite wall. PIVs of these velocity data are constructed and compared to the predictions of CFD simulations of varying fidelity.

11:21AM HX.00005 Zones of Influence and Low Frequency Shock Motion in a Shock Boundary Layer Interaction

LIONEL AGOSTINI, LIONEL LARCHEVEQUE, PIERRE DUPONT, JEAN-FRANCOIS DEBIEVE, JEAN-PAUL DUSSAUGE, IUSTI, UMR 6595 CNRS / University of Provence — Shock-wave / boundary layer interactions usually exhibit unsteadiness with strong low frequency content. The present work aims at analyzing the low frequency shock motion using high resolution data from long-time large-eddy simulations. Three different flow configurations are considered yielding incipient to full separation of the boundary layer in the interaction region. Filtered cross-correlation are used to identify the flow regions being able to influence the shock at low-frequency. It is demonstrated that the information paths deduced from the cross-correlations coincide with the pressure characteristic lines. A theoretical computation of the phase velocity along the shock of perturbations induced by the "breathing" of the interaction region is derived. For all the three flow configurations, two different velocities are found, depending on the location of the sources along the boundary of the decelerated zone. These velocities match quite accurately velocities along the shock computed from the LES data by means of cross-correlations.

11:35AM HX.00006 Influence of wall heating on a shock-wave / turbulent boundary layer interaction

LIONEL LARCHEVEQUE, JEAN-PAUL DUSSAUGE, IUSTI, Provence University and CNRS, Marseille, France — Shock / boundary layer interactions, if strong enough, are known to result in separation of the incoming boundary layer and to exhibit strong unsteadiness. The main purpose of this work is to study the influence of wall heating on these features by means of Large-Eddy Simulations. Six LES have been carried out for two shock angles, either with adiabatic or heated lower walls. The unsteady data thus obtained will be used to evaluate the effects of wall temperature fluxes on the separation size, the separation state and the unsteady behavior of the separated flow at low and medium frequencies. The influence of the wall heating on the three-dimensional modulations found within the separation bubble, either intrinsic or induced by side walls, will also be analyzed. Computations taking into account side boundary layers will particularly be used to demonstrate that lower wall heating is an effective way to achieve experiments exhibiting large separation bubbles with restricted influences of the wind tunnel side walls.

12:01PM HX.00008 A-priori and a-posteriori assessment of SGS models for shock-boundary layer interactions

AVINASH JAMMALAMADAKA, ZHAORUI LI, FARHAD JABERI, Michigan State University — A-priori and a-posteriori assessments of subgrid-scale (SGS) large-eddy simulation (LES) models are made for an incident shock wave interacting with a Mach 2 flat-plate supersonic turbulent boundary layer using direct numerical simulation (DNS) data. The governing equations for DNS and LES are solved using the seventh-order Monotonicity Preserving scheme for Euler flows and the sixth-order compact scheme for viscous terms. The SGS models tested included constant coefficient and dynamic eddy-viscosity and similarity models. A-priori tests confirm that the similarity- and mixed-type models are superior to those developed based purely on eddy-viscosity assumption. However, some of the eddy-viscosity models still perform adequately in a-posteriori tests. Overall, dynamic models show reasonably good agreement with the DNS data.
12:14PM HX.00009 DNS of shock-turbulent boundary layer interaction, SUMAN MUPPIDI, KRISHNAN MAHESH, University of Minnesota — A novel DNS/LES capability for high speed viscous flows on unstructured grids is being developed. Shock-turbulent boundary layer interaction results in flow separation, shock unsteadiness, and increased aerodynamic and thermal loads. This paper focuses on the DNS of a Mach 3 turbulent boundary layer flow past a 24 degree compression corner. In our simulations, the upstream turbulent boundary layer is obtained by roughness-induced transition of a laminar boundary layer, and not by rescaling methods. The simulations are performed at the conditions of experiments by Bookey et al (AIAA Paper 2005). The results will be compared to experimental data. We will present the evolution of the boundary layer flow across the shock, low frequency unsteadiness, and the upstream influence of the corner. Both numerical issues and their physical implications will be discussed.

12:27PM HX.00010 Similarity scaling in shock-turbulence interactions, DIEGO DONZIS, TEXAS A&M University — The interaction of turbulence with a normal shock is an important problem of both fundamental and practical interest. While some trends are relatively established (e.g., turbulent kinetic energy amplification) quantitative predictions remain elusive when different flow conditions are considered. Data are typically compared with the Linear Interaction Approximation (LIA, Ribner 1953) which neglects the influence of Reynolds and turbulent Mach numbers, though data show that their effect is significant at conditions achievable in simulations and experiments. Without a suitable criterion to identify different regimes including what constitutes asymptotic states (e.g., high Reynolds number) one can come to varying conclusions from simulations and experiments. Similarity scaling is used to analyze available data of isotropic turbulence interacting with a normal shock wave. The proposed analysis suggests that incomplete similarity occurs for several parameters as opposed to complete similarity assumed in LIA. A combination of similarity parameters related to the shock thickness is found to be key even when it is small and, therefore, cannot be neglected. Within this theoretical framework, data on e.g., amplification factors and shock structure are found to present universal behavior in the proposed parameter which allows for identification of different regimes. Simple models based on these results can capture the scaling of the shock structure. Further implications of findings will be discussed.

Monday, November 22, 2010 10:30AM - 12:40PM — Session HY Instability: Interfacial and Thin Film V Hyatt Regency Long Beach Regency E

10:30AM HY.00001 Marangoni-Bénard instability in microgravity: PTV-analysis of the velocimetry data generated during the BAMBI - FOTON-M2 experiment1, SAM DEHAEC, SAMUEL TALVY, ALEXEY REDNIKO, Université Libre de Bruxelles, TIPs laboratory, PATRICK QUEECKERS, Université Libre de Bruxelles, MRC laboratory, PIERRE COLINET, Université Libre de Bruxelles, TIPs laboratory — The BAMBI (Bifurcation Anomalies in Marangoni-Bénard Instabilities) experiment has been successfully flown onboard the FOTON-M2 satellite in June 2005. During the 4 days available for the experiment, a 5mm-thick 200 cSt silicone oil layer in a 10x10cm² wide container, and in contact with a similarly-sized helium gas layer was heated from "below" and cooled from "above." By varying the heating power applied at each experimental step, a range of temperature differences across the liquid and gas layers was scanned and the onset and evolution of the Marangoni-Bénard instability typical for this type of configuration was examined. The used optical diagnostics were Infrared Thermography of the liquid/gas interface, PTV (multiple views and heights in the liquid layer), Wollaston Interferometry and Electronic Speckle Pattern Interferometry. The present contribution focuses on the velocity results obtained by PTV in the interface plane, and discusses them in relation both with infrared images, and with theory/numerics.

1Supported by ESA & BELSPO PRODEX projects and by FRN-FNRS.

10:43AM HY.00002 Experimental analysis of evaporation-driven Bénard instabilities1, FABIEN CHAUVET, SAM DEHAEC, PIERRE COLINET, Laboratory TIPs (Transfers, Interfaces and Processes), Fluid Physics Unit, Université Libre de Bruxelles, Belgium — We study experimentally the spontaneous patterns induced by evaporation of a pure liquid layer into dry air. The liquid/vapour interface temperature is lower than the substrate temperature because of the energy consumption for the phase change. This temperature difference across the liquid layer generates surface-tension-driven convection and/or buoyancy-driven convection in the liquid, depending on the layer thickness. In practice, the volatile liquid is placed in a circular dish placed in dry air under ambient conditions. During the evaporation process, the convective patterns are observed using an optical Z-type Schlieren set-up. The evaporation rate and the layer thickness are estimated from liquid weight measurements using a precision balance. Several liquids and dish diameters/heights have been tested. As could be expected, the evaporation rate remains almost constant during time, while different convective patterns are observed when the layer decreases. Their phenomenology and transitions are analysed, in relation with existing theoretical models.

1Supported by the Marie Curie MULTIFLOW Network, by ESA & BELSPO PRODEX projects, and by FRN-FNRS.

10:56AM HY.00003 Suppressing van der Waals driven rupture through shear1, MICHAEL DAVIS, MICHAEL GRATTON, STEPHEN DAVIS, Department of Engineering Sciences and Applied Mathematics, Northwestern University — A thin viscous film on a substrate is susceptible to rupture instabilities driven by van der Waals attractions. When a sufficiently large shear is applied to the free surface, the rupture instability is suppressed in two dimensions for sufficiently large shear magnitude and replaced by a permanent finite amplitude travelling wave with speed approximately equal to the speed of the surface. For small amplitudes, the wave is governed by the Kuramoto-Sivashinsky equation. If three-dimensional disturbances are allowed, the shear is decoupled from perpendicular disturbances to the flow, and line rupture would occur. In this case, replacing the unidirectional shear with a rotating shear can suppress rupture for suitable choices of shear magnitude and angular velocity.

1National Science Foundation, Grant Nos. CMMI-0826703 (MJD, SHD) and DMS-0636574 (MBG).

11:09AM HY.00004 Experimental Confirmation of Pillar Array Formation in Polymer Nanofilms by Thermocapillary Instability, EUAN MCLEOD, YU LIU, SANDRA TROIAN, California Institute of Technology, 1200 E California Blvd, MC 128-95, Pasadena, CA, 91125 — During the past decade, three mechanisms have been proposed to explain the spontaneous formation of periodic fluid elongations in polymer nanofilms confined to closely spaced parallel substrates held at different temperatures. Models suggest linear instability due either to variation in surface charge density at the air/polymer interface, variation in acoustic phonon pressure within the film, or variation in thermocapillary stress along the air/polymer interface. Comparison of theory to experiment requires that the film structuring process be observed at early times in accordance with the assumptions of linear stability analysis. To date, however, all experimental investigations of the most unstable wavelength have been conducted in the solidified state and long after the original molten structures had contacted and reorganized along the cooler substrate. For the first time, we present experimental measurements based on direct observation of in-situ nanopillar growth. Investigation of the fastest growing wavelength as a function of substrate separation distance, temperature difference and initial film thickness indicates excellent agreement with predictions of the thermocapillary model. These studies also make evident how film depletion and contact may have skewed values of the wavelength previously reported in the literature.
Viscous dispersion effects on bound-state formation in falling liquid films

Marc Pradas, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK; Dmitri Tseluiko, School of Mathematics, Loughborough University, Leicester, LE11 3TU, UK; Serafim Kalliadasis, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — We examine the influence of viscous dispersion on the interaction of two-dimensional solitary pulses in falling liquid films at moderate Reynolds number. We make use of an averaged model that includes second-order viscous effects in the long-wave expansion. These effects play a dispersive role affecting primarily the stability and the dynamics of the solitary pulse. We show that different dispersion parameters, such as surface tension and viscosity, play a crucial role in the interaction between pulses giving rise eventually to the formation of bound states consisting of two or more pulses separated by well-defined distances and travelling at the same velocity. By developing a coherent-structures theory that assumes weak interaction between the pulses, we are able to theoretically predict the pulse-separation distances for which bound states are formed. It is shown that viscous dispersion significantly affects the distances at which bound states are observed. In all cases, there is very good agreement between the theory and computations of the fully nonlinear system.

Fingering instability down the outside of a vertical cylinder

Linda Smolka, Bucknell University, MARC Segall, University of Central Florida — We present an experimental and numerical study of the dynamics of a gravity-driven contact line of a thin viscous film traveling down the outside of a vertical cylinder of radius R. In all of our experiments with cylinder radii ranging between 0.159 and 3.81 cm, the contact line becomes unstable to a fingering pattern. Observations are compared to inclined plane experiments in order to understand the influence curvature plays on the fingering pattern. Using lubrication theory, we derive a model for the film height that includes gravitational and surface tension effects and examine the structure and linear stability of the contact line using traveling wave solutions. For Bo ≥ 0(10^4), our model predicts curvature’s influence is negligible as the shape and stability of the contact line converge to the behavior one observes for a vertical plane. For Bo > 1, the most unstable and cutoff wave modes and maximum growth rate scale like Bo^{0.45}, indicating instability of the contact line increases as gravitational effects increase or, for a specific fluid, as cylinder radius increases. The linear stability of the contact line changes at the critical value Bo_c = 0.56; above Bo_c, the contact line is unstable and below Bo_c, it is stable to fingering. We find excellent agreement between the number of fingers that form along the contact line and the range of wavelengths measured in experiments and the range of unstable modes and wavelengths predicted by our model.

Transverse Fracture Bands during Rapid Peeling of Adhesive Tape

S.T. Thordssen, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia; H.D. Nguyen, T. Takahara, T.G. Etoh, Kinki University, Osaka, Japan — The typical roll of Scotch tape consists of sticky acrylic adhesive attached to an acetate film, with a total thickness of 58 µm. When this adhesive tape is peeled from a solid surface, the detachment occurs with a well-known stick-slip mechanism accompanied by a characteristic ripping sound. Here we present direct ultra-high-speed video imaging of the detachment zone when Scotch tape is peeled off at high speed from a solid glass surface. The tape is manually pulled from the surface at very large velocities between 4 - 14 m/s and is viewed through the substrate, with a long-distance microscope. The video imaging at 1 million fps reveals a highly regular substructure of transverse fractures, which appear during the slip phase. The typical 4 mm-long slip region has a regular substructure of transverse 220 µm wide slip bands, which fracture sideways at speeds over 300 m/s. Our imaging can observe the growth and relaxation of cavitation bubbles within the adhesive layer. The fracture tip emits waves, which travel up the detached section of the tape at ~100 m/s. We believe this promotes the sound, so characteristic of this phenomenon.

Influence of Inertia, Gravity and Thermal Conditions on the Draw Resonance

Zheming Zheng, Olus Boratav, Corning Incorporated — The instability known as the “draw resonance” is studied for a Newtonian viscous flow. Both eigenvalue analysis and transient solutions are used to study the instability. The effects of inertia, gravity and the thermal conditions on stability are explored. The thermal conditions are studied as a combination of global and local heating/cooling conditions. While monotonous global cooling along the draw always gives critical draw ratios less than that of the isothermal case (i.e. Dr* = 20.218 for isothermal), critical draw ratios larger than 20.218 can be obtained by local heating effects superposed onto the global cooling. The global heating is stabilizing and very large critical draw ratios are obtained when the intensity of this global heating is large.

Gravity-driven propagation of thin non-isoviscous rivulets on vertical and inclined planes

Andrey Filippov, Gaozhu Peng, Corning Incorporated — Many practical problems require the spreading of a liquid on a solid. In the glass industry, flows of molten glass on a vertical or inclined, in respect to the vertical, solid refractory surface are parts of several important applications. In present paper, propagation of a thin and relatively narrow rivulet on vertical and inclined solid planar surface is considered within a mathematical frame of general lubrication theory, using a commercially available PDE solver. In contrast to most of previous studies, the addressed flows are gravity driven, and the coefficient in front of the surface tension term in the dimensionless equations (the inverse Bond number) is small and does not exceed 10^{-5}. It has been found that the flow pattern strongly depends on the inclination angle. For example, the contact line of rivulets propagating on vertical and negatively inclined planes behaves similarly; it leads one or several smaller footprints of a vertical cylinder. In all of our experiments, we have found that gravity is the most important factor, followed by inertia. The effect of gravity increases with gravity and inertia, and the effect of inertia is less important than the effect of the gravitational force.

Effect of Variable Viscosity on Stabilization in Hele-Shaw Flows

Prabir Daripa, Texas A&M University — Current work is in progress on numerical investigation of the effect of variable viscosity profiles of internal layers on the stabilization of multi-layer Hele-Shaw flows. Our findings will be presented and compared, to the extent possible, with theoretical results available. Effect of diffusion on this stabilization may also be presented. This talk is partially based on an ongoing work with Xuexu Ding.

Supported by the Qatar National Research Fund under award # 481010.

Monday, November 22, 2010 10:30AM - 12:27PM

Session HZ Instability/General II

Hyatt Regency Long Beach Regency F

10:30AM HZ.00001 Dynamics of Coherent Structures in Localized Turbulence in a Pipe

Jerry Westerweel, Dirk-Jan Kuij, Delft University of Technology — The transition to turbulence in pipe flow is still not completely understood. Recently it was shown that localized turbulent structures (puffs) can survive for hundreds of pipe diameters (or integral time scales) and then suddenly disintegrate. Questions that emerge are: Why is the turbulence localized? What mechanism is required for puffs to sustain itself? What changes in the structure of a puff when it suddenly decays? For the investigation a high resolution DNS is used. The high resolution is required to resolve the localized high energy peaks, which were observed in earlier experimental investigations. We use a stereoscopic planar PIV measurement as initial condition for the DNS and continued the time evolution at Re=1900. The first observation is that the velocity of the structures is higher than the bulk velocity at Re=1900 as opposed to the Re=2500 case, which is in agreement with experimental observations. The peaks in in-plane kinetic energy are reproduced in the DNS, and can be associated with hair-pin vortices.
10:43AM HZ.00002 Spatiotemporal chaos and infinite-lifetime turbulence in pipes and channels\textsuperscript{1}, DWIGHT BARKLEY, DAVID MOXEY, University of Warwick — Lifetime measurements of localized states have been the focus of many recent studies of transitional turbulence. We argue that the transition to infinite-lifetime turbulence must be understood as a transition to spatiotemporal chaos, similar to directed percolation (although the transition may not be strictly DP). While such arguments were first made many years ago, we report evidence substantiating this view from direct numerical simulations of long pipes. We also report work on modeling these phenomena.

\textsuperscript{1}Work supported in part by the Leverhulme Trust and the Royal Society. Computational support came from IDRIS.

10:56AM HZ.00003 Global instabilities of the flow over a backward-facing step, DANIEL LANZER-STORFER, HENDRIK C. KUHLMANN — The three-dimensional linear stability of the two-dimensional, incompressible flow over a backward-facing step is considered. The geometry is varied covering an expansion ratio from 0.091 to 0.975. The basic flow becomes unstable to three different three-dimensional modes depending on the expansion ratio. An energy-transfer analysis is used to understand the nature of the instability. In the limit of vanishing step height the critical mode is stationary and the amplification process is caused by a Kelvin-Helmholtz-type instability. For high expansion ratios the basic flow features a wall-jet structure and becomes unstable due to centrifugal forces with respect to an oscillatory mode. For intermediate expansion ratios an elliptic instability mechanism is identified and the instability characteristics change continuously with the expansion ratio.

11:09AM HZ.00004 Structural changes of laminar separation bubbles induced by global linear instability\textsuperscript{1}, DANIEL RODRIGUEZ, California Institute of Technology, VASSILIS THEOFILIS, School of Aeronautics, Universidad Politecnica de Madrid — Global modal linear instability analysis considers three-dimensional disturbances superimposed upon (essentially non-parallel) two- or three-dimensional basic flows. Here two-dimensional (BIGlobal) analysis of laminar separation bubbles embedded in a flat-plate boundary layer is performed. Results obtained show the presence of a stationary three-dimensional eigenmode, which is unstable for a finite range of spanwise wavenumbers, while the same steady basic flow is stable against two-dimensional disturbances of the Kelvin-Helmholtz/Tollmien-Schlichting class. Critical-point theory shows that 2D flow is "structurally unstable" and the presence of any 3D disturbance, like the aforementioned global mode will alter the complete topological description regardless of the disturbance amplitude. Critical-point theory is used here in order to characterize the different topological bifurcations exerted by global instability on the steady laminar two-dimensional bubble: a spanwise modulation of the separated region appears, eventually leading to the breakdown of the recirculation region into independent cellular structures, highly reminiscent to the patterns observed experimentally on stalled airfoils.

\textsuperscript{1}Based on work sponsored by AFOSR/EOARD Grant FA8655-06-1-3066.

11:22AM HZ.00005 Pinning of rotating waves in systems with imperfect $SO(2)$ symmetry, FRANCISCO MARQUES, ALVARO MESEGUER, Universitat Politècnica de Catalunya, JUAN M. LOPEZ, RAFAEL PACHECO, Arizona State University — Experiments in small aspect-ratio Taylor-Couette flows have reported the presence of a band in parameter space where rotating waves become steady non-axisymmetric solutions (a pinning effect) via infinite-period bifurcations that previous numerical simulations were unable to reproduce. Here we present numerical simulations that include a small tilt of one of the endwals, simulating the effects of imperfects that break the $SO(2)$ axisymmetry of the problem, and indeed are able to reproduce the experimentally observed pinning of the rotating waves. A detailed analysis of the corresponding normal form shows that the problem is more complex than expected, and the complete unfolding is of codimension six. A detailed analysis of different types of imperfections indicates that a pinning region surrounded by infinite-period bifurcation curves appears in all cases. Complex bifurcational processes, strongly dependent on the specifics of how the symmetry is broken, appear very close to the intersection of the Hopf bifurcation and the pinning region. The numerical and theoretical results agree with the previous experimental studies.

11:35AM HZ.00006 The attractor manifold of the flow past a circular cylinder for $Re = 100^1$, IAGO C. BARBEIRO, JULIO R. MENEGHINI, J.A.P. ARANHA, NDF, Escola Politecnica, University of Sao Paulo, Brazil — The flow past a circular cylinder in its two-dimensional nonstationary régime is concerned, in the vicinity of $Re = 100$. At this point it shows the behavior of a self-sustained oscillator with a simple attractor, the periodic solution. This study proposes a methodology to build the attractor manifold from one picture of the solution inside the attractor simple attractor, the periodic solution. This study proposes a methodology to build the attractor manifold from one picture of the solution inside the attractor.

\textsuperscript{1}Authors have grants from FINEP-CTPetro, FAPESP and Petrobras.

11:48AM HZ.00007 Experiments in the stability of basic two-dimensional flows\textsuperscript{1}, PAUL W. FONTANA, EDWARD C. TITMUS, ADRIAN KIRN, Seattle University — Two-dimensional flows have different stability behavior than their three-dimensional counterparts due to enstrophy conservation, but they have not been studied as thoroughly in experiments. We present data from quasi-two-dimensional flow experiments suggesting that basic shear flows should exhibit instability not predicted by theory, while square-root-lattice flows are more stable than predicted by linear theory. To allow proper quantitative comparisons between experiments and theory we have developed new techniques for quantifying and distinguishing kinematic viscosity and Ekman friction.

\textsuperscript{1}Supported by the National Science Foundation under Grant No. CBET-0854509 and the M. J. Murdock Charitable Trust.

12:01PM HZ.00008 Measurement Techniques: Viscosity and Surface Drag in Quasi-Two-Dimensional Flows\textsuperscript{1}, EDWARD C. TITMUS, ADRIAN T. KIRN, PAUL W. FONTANA\textsuperscript{2}, Seattle University — The effects of kinematic viscosity and linear drag are both significant in many quasi-two-dimensional (Q2D) flows in nature and the laboratory. These effects, however, are difficult to measure and to distinguish from one another. We demonstrate precise, independent measurement of both kinematic viscosity and linear drag using decay rates of vortices of varying scales in a Q2D experiment involving soap films in a circular Couette cell. As expected, we have found both the kinematic viscosity and the linear drag to depend inversely on film thickness. The approach can be generalized to apply to other configurations and experiments.

\textsuperscript{1}Supported by Seattle University, the National Science Foundation under Grant No. CBET-0854509, and the M. J. Murdock Charitable Trust.

\textsuperscript{2}[PI]
12:14PM HZ.00009 Experimental Exploration of the Dispersion Relation of Rayleigh-Taylor Instability, MARIE-CHARLOTTE RENOUFT, CHIA-LING CHEN, SAMEH FERJANI, PIERRE CARLÈS, CHARLES ROSENBLETT, Case Western Reserve University — Investigating arbitrary initial configurations in the Rayleigh-Taylor instability presents an experimental challenge due to the difficulty controlling the initial interface shape. To overcome that, we pioneered in 2006 the use of magnetic levitation. In our current set-up, the denser paramagnetic fluid is levitated above the lighter fluid, using a quasi-homogeneous magnetic force. In order to modulate the static interface shape as desired, magnetically permeable elements such as straight pieces of magnetically permeable wire are added in the device. The initial interface thus is no longer flat and the destabilization of the interface is observed after the magnetic field is turned off. We will show our recent results of a systematic exploration of the dispersion relation of the instability by using straight segments of wires of different lengths. We will concentrate on the linear growth rate calculation with two different independent approaches and point out the consistency of the two methods. Beyond the verification of a known theory, this first attempt paves the way for a new systematic exploration of non-linear growth and mode couplings in the Rayleigh-Taylor instability for more original distributions of initial perturbations.

Monday, November 22, 2010 2:00PM - 2:35PM — Session KT Invited Session: Dynamics of Suspended Colloidal Particles Near a Wall: Electrokinetic Effects and Implications for Particle-based Velocimetry Long Beach Convention Center Grand Ballroom B

2:40PM KT.00001 Dynamics of suspended colloidal particles near a wall: Electrokinetic effects and implications for particle-based velocimetry, MINAMI YODA, Georgia Institute of Technology — Interfacial phenomena due to surface forces are important in microfluidic devices with their relatively large surface areas and small volumes, and may lead to new ways to control microscale flows. Most experimental studies of interfacial transport estimate flow velocities from the motion of “nanoparticles,” e.g. polystyrene (PS) beads and quantum dots, and assume that the particle displacements over a known interval are the fluid velocities. This talk discusses some of the challenges in extracting fluid velocities within about 500 nm of the wall from the motion of fluorescent PS spheres as small as 100 nm illuminated by evanescent waves. Because the evanescent-wave intensity decays exponentially with wall-normal distance, the particle-wall separation can be determined from the brightness of each particle image. Unlike a similar technique, total internal reflection microscopy, which usually studies the dynamics of a single colloidal particle in a quiescent fluid, this work considers instead an ensemble of O(10^4) particles convected by a flow. For steady Poiseuille and electrokinetically driven flows through channels less than 50 μm deep, the distribution of this ensemble of colloidal particles near the wall is strongly nonuniform, due mainly to repulsive particle-wall electric double layer interactions and van der Waals forces. An electric field parallel to the wall changes the particle distribution by creating an additional “electrokinetic lift” force that depends upon particle and fluid properties as well as the electric field magnitude. After accounting for this nonuniform distribution, the diffusion coefficients and the fluid velocities extracted from the particle displacements are in agreement with theoretical predictions. Due to electrokinetic lift, particle displacements in a shear flow could vary for different tracers, and give different velocity results unless corrected for variations in particle distribution.
L1.00001 Particle transport near arterial stenosis. SAHAR HENDABADI, SHAWN SHADDEN, Illinois Institute of Technology — We will present work towards understanding particle transport near arterial stenoses. Prior studies have shown increased platelet aggregation downstream of stenosis, or analogous geometrical models that induce flow separation and recirculation via abrupt expansion. Stenosis leads to changes in fluid mechanical quantities such as shear stress, flow separation, recirculation and reattachment and there exists several hypotheses on how these conditions influence platelet activation and aggregation. In particular, it is thought that high shear at the stenotic throat “activates” platelets that subsequently aggregate in the low shear separation zone perpetuating thrombotic events. We aim to understand particle (e.g. platelet) transport downstream of a stenosis in close detail. Towards this objective, we have developed numerical models of pulsatile flow near arterial stenoses and methods for particle tracking, including quantification of mechanical stimuli thought to initiate platelet activation. We will discuss results of this effort, comparison with previous studies, and plans for continued numerical and experimental work.

L1.00002 Toroidal Lagrangian Flow Structures in highly viscous fluids by moving bent rods1, PAVEL CHTCHEPROV, ROBERTO CAMASSA, DAVID HOLZ, DAVID MARRON, JAMES MARTINDALE, RICHARD MCLAUGHLIN, LEANDRA VICCI, LONGHUA ZHAO, University of North Carolina, UNC NSF RTG FLUIDS GROUP COLLABORATION — Motile cilia play a large role in fluid motion across the surface of ciliales. Flows caused by the cilia move debris and mucus through mass beat patterns controlled by the motor proteins while rotating about the basal body that attaches the cilia to the cell surface. This study approximates the cilia as a slender body rotating about a point of contact of one of its ends in a viscous fluid. The bent rod sweeps out a virtual cone with a chord connecting both ends. The bend of the rod, the cone angle, the angle between the central axis to the normal plane, and the angle of rotation of the bent rod about its chord affect the flow patterns in a Stokes fluid. The slender body theory allows for an asymptotic solution of the Lagrangian trajectories and flow patterns caused by the precessing rod, which can be directly compared to experimental data. Altering the above parameters produces different toroidal flow structures. Using 3D stereo calibration, accurate quantified comparisons of epicyclic particle trajectories in short and long time are made against the model predictions.

1 NSF

L1.00003 Mechanical response of solutions of phospholipids to anisotropic compression1, YVES DUBIEF, School of Engineering, University of Vermont, Burlington, VT, LEONIE COWLEY — Solvated phospholipids in concentration higher than the critical micellar concentration self-assemble in micelles, vesicles and/or bilayer membranes. These structures are the building blocks of many biological systems, such as cell membranes and lining of lungs, and are well known for their ability to provide a biophysical barrier between two mediums. Another property that has not received as much attention is their macroscopic mechanical role based on nanoscale interactions, which is hypothesized to play a major role in the functions of articular joints. Using atomistic and coarse-grained molecular dynamics (MD), the existence of two universal linear elastic regimes of multilamellar bilayer membranes under anisotropic compression is identified for different level of hydration. Leveraging this property, the repulsive dynamics between self-assembled structures and the reduction of diffusion in supported membranes, a fluid nano ball bearing is constructed as an illustration for our hypothesis of synovial lubrication.

1 Computational time provided by the Vermont Advanced Computing Center at the University of Vermont.

L1.00004 Effect of the interaction on growing / condensation processes of vapor bubbles injected in subcooled pool, RYOTA HOSOYA, Div. Mechanical Engineering, Grad. School of Science & Technology, Tokyo University of Science, ICHIRO UENO, Dept. Mechanical Engineering, Fac. Science & Technology, Tokyo University of Science — We carry out an experimental study with a special interest on a growing and collapsing processes of vapor bubbles injected into a subcooled pool. In the present system, we extract the liquid-vapor interaction in the boiling phenomenon consisting of complex three-phase interactions. Vapor of distilled water is generated in the vapor generator apart from the pool, and then is injected to the pool at a designated degree of subcooling. The degree of subcooling of the pool is controlled from 8 to 80 K. Bubble growth and condensation processes are detected by a high-speed camera at frame rate up to 140,000 fps with backlight illumination. We successfully detect an instability arising on the vapor bubble interface in prior to the abrupt condensation to collapse. We figure out occurring condition of such instability by evaluating a condensing rate as functions of the degree of subcooling and the vapor injection rate. We then pay our special attention to the interaction of adjacent vapor bubbles injected through two neighboring orifices. Effect of interaction between vapor bubbles is discussed considering a distance between the orifices, and the degree of subcooling.

L1.00005 Shock Turbulence Interaction using Observable Euler Equations, CHANG XIAO, KAMRAN MOHSENI, University of Colorado at Boulder — Accurate numerical simulations of turbulent flows requires minimizing numerical dissipation, while the usual shock-capturing schemes need numerical dissipation for the algorithm stabilization. To overcome this dilemma, the observable Euler equations (Mohseni, AIAA paper 2009-6695) were proposed as a technique for simultaneous regularization of shocks and turbulence. The effects of the observable Euler equations in 3D shock turbulence simulation have been tested in several problems including the Shu-Osher, Taylor-Green Vortex, Noh problem, and decaying compressible isotropic turbulence with eddy shocklets. The Taylor-Green Vortex problem tests the stability for severely under-resolved motions, as well as a measure of the preservation of kinetic energy and the growth of enstrophy. In the Noh problem, strong shock waves interact with interfaces separating different fluids and with the resulting turbulence. It tests the capability to handle a strong spherical shock. In the case of decaying compressible isotropic turbulence with eddy shocklets the ability of the observable Euler equations to handle “randomly” distributed shocklets, as well as the accuracy for broadband motions in the presence of shocks was tested.

L1.00006 Transport of an Active Scalar in a Chaotic Flow Field1, HIMANSHU TIWARI, MARK PAUL, Virginia Tech — We study the active transport of a scalar species in the flow field of Rayleigh-Bénard convection that is exhibiting spatiotemporal chaos. Recent work has quantified the passive transport of a scalar species in a spiral defect chaos flow field to yield enhanced diffusion. In this work we are interested in allowing the scalar species to undergo active transport. For example, the combustion of premixed gases where the scalar quantity can react, or the motion of microorganisms in bioconvection where the scalar quantity can swim in some preferential direction. We use large-scale numerical simulations to solve a reaction-advection-diffusion equation for the scalar species simultaneously with the three dimensional time dependent Boussinesq equations. We use our numerical results to explore the transport of an active scalar species in a variety of chaotic flow fields and over a range of Lewis numbers.

1 NSF

L1.00007 ABSTRACT WITHDRAWN —
The safety of a cell in a droplet under high electric field\textsuperscript{3}. JIHOON NOH, DO DIN IM, IN SEOK KANG, POSTECH — Electrically charged aqueous droplet can be transported by electrical field in a dielectric fluid without the flow of medium (Jung et al. J. Colloid Interface Sci. 2008). This phenomenon can be used to transport a single nanoliter droplet in a micro channel, which can serve as biochemical micro-reactor. Because an aqueous droplet is much conductive than the dielectric fluid, there is effectively no electric field inside the droplet suspended in dielectric fluid. Therefore bio-materials are protected from electricity even under high electric field. However, when the droplet is charged near an electrode by direct contact to the electrode, there is possibility that electric field can hurt bio-materials like DNA molecules, microorganisms, cells, protein in droplet. Because of this concern, we should confirm that bio-materials in droplet moving by direct charging are safe under strong external electric field especially to organism cells. Therefore we examine the effect of electric field on the cells such as yeast, E.coli., and sperm in droplet experimentally.

Behavior of contact angles on rough solid surfaces. YUMIKO YOSHITAKE, Science and Engineering, Tokyo Denki University, KO OKUMURA, Department of Physics, Ochanomizu University — In this study, we consider a sinuousoidal surface and show explicitly how the pinning and depinning occur for a two dimensional liquid drop on such non-ideal surfaces as the volume of the drop is increased or decreased. The surface energy of the drop have a wave like form affected by solid surface shapes and contact lines are pinned by the energy barriers. We show that the contact angle hysteresis (CAH) emerges from this simple model even though we do not take any effect of viscous dissipation into account.

Study of drop coalescence using the Lattice-Boltzmann Method. RICARDO FALLA AVILA, ANDRES GONZALEZ-MANCERA, Universidad de los Andes — Drop coalescence in an emulsion is studied using the Lattice-Boltzmann Method (LBM) to simulate multicomponent fluid flow in 3D. Two cases are considered, drop interaction under shear flow and interaction during gravity induced sedimentation. Several experimental results on the topic have already been published and computational studies using the Boundary Element Method (BEM) are also available, however, it is of interest to investigate how well does the LBM behaves in comparison with other methods. For shear flow, drop size ratios, interfacial properties and flow regimes, which favor coalescence, are identified and discussed. Multiple drops interactions under shear flow are demonstrated and characterized. The interaction of drops under gravity-induced sedimentation is also considered. In this case, the effects of drop size ratio, surface tension, gravitational pull and viscosity ratio on drop interaction and coalescence are studied. The results are then compared to those obtained using the traditional BEM and experimental data from the literature. Discussion on code implementation, as well as advantages and disadvantages of each method are highlighted.

Modern applications of shadowgraph imaging\textsuperscript{4}. RAFAEL CASTREJÓN GARCÍA\textsuperscript{2}, Universidad Nacional Autónoma de México. JOSE RAFAEL CASTREJÓN-PITA\textsuperscript{1}, GRAHAM D. MARTIN, IAN M. HUTCHINGS, University of Cambridge — Over the last hundred years the shadowgraph technique has been extensively used in the study of fluid dynamics, the visualization of objects in motion and in the optical inspection of transparent media. Shadowgraphy is often considered an inexpensive but powerful tool to visualize liquids and is generally used to obtain qualitative properties, such as shapes and motion behavior of gases and liquids. When the shadowgraph technique is combined with digital image analysis generate quantitative data. Three experimental systems are described. The first example is a setup developed to visualize and quantify the droplet size distribution in sprays. The second is a shadowgraph system used to record and analyze the profile of modulated continuous jets in order to measure the dynamic surface tension via the Raleigh-Weber model. The final system uses shadowgraph images of moving droplets to identify and record their instantaneous position and direction of motion.

PIV measurements of flow characteristics induced by mini plate-wing plasma actuators. A.N.M. MOMINUL MUKUT, PhD Student, HIROSHI MIZUNUMA, Professor, TAKEHIKO SEGAWA, Researcher, OBARA HIROMICHI, Associate Professor — The surface DBD plasma actuator is known to be effective for flow control process. Plasma is produced on actuator and gives a body force to the ambient air which is the mechanism for active flow control. Until now, the actuators have been mounted on the wall surface. The plasma actuator is thin and controllable electrically. If we combine the plasma actuator and the passive devices like a vortex generator and Large Eddy Break Up device, those passive devices would be activated. As the basis of the combination use, this paper investigated the wing-like plasma actuators, the width and chord length of which were 96mm and 19.6mm respectively. The electric wind was generated in the absence of external flow by the plasma actuator. Two electrodes were separated by a Kapton thin wing plate and were located at 5.75mm or 14mm from the leading edge. The induced flow was compared as a function of the distance from the leading edge to the actuator position. It was found that the increase in the distance shifted the point of maximum velocity downstream but the induce wake flow indicated the same momentum integral.

Reduced-Order Models of a Natural Convection Loop for Known Heat Flux Conditions via Karhunen-Loève Expansions\textsuperscript{1}. TOBIAS HUMMEL, ARTURO PACHECO-VEGA, California State University, Los Angeles — We build reduced-order dynamical models of a thermal convection loop using the Karhunen-Loève decomposition (KL) methodology, in conjunction with the Galerkin projection technique. The convective loop has the form of a torus and is filled with a water. The loop receives heat in some parts and releases it in others through a known-heat-flow sinusoidal function, thus creating a natural circulation. Under suitable assumptions, the momentum and energy equations are reduced to a set of one-dimensional integro-differential equations, in which the tilt angle of the loop and the heat rate per unit length are the bifurcation parameters. The set of equations is first solved via finite differences to generate numerical solutions from which the KL model can be built. Then, the method of snapshots and the Galerkin projection are applied to find the KL basis functions, and the corresponding constants, that generate the most compact dynamical system. It is found that the number of KL modes required to build a model is a function of the linear stability of the steady states. As the system goes from stable to unstable regions, and finally to chaos, the number of required modes increases. However, for an accuracy level of, e.g., 10\textsuperscript{-4}, these reduced-order models are at five orders of magnitude faster than the finite difference solver.

Investigation of Internal Wave Spectra due to Observed Interactions. BENJAMIN HILLYARD, JULIE VANDERHOFF, Brigham Young University, Provo — Fluids such as the ocean and atmosphere are stably stratified such that the density within the fluid increases with depth. Perturbations to the stratification for example by flow over topography, convective storms or turbulent mixing can lead to the generation of internal waves. As these waves are generated and propagate through their respective media, they interact with a multiplicity of other internal waves. Each wave-wave interaction outcome is governed by the parameters of each wave involved in the interaction. When one of the waves is significantly larger scale, such as an inertial wave, linear theory may be used to assess the interaction between it and a smaller scale wave. The result is three basic types of interactions: wave-wave interaction outcome is governed by the parameters of each wave involved in the interaction. When one of the waves is significantly larger scale, such as an inertial wave, linear theory may be used to assess the interaction between it and a smaller scale wave. The result is three basic types of interactions: wave-wave interaction outcome is governed by the parameters of each wave involved in the interaction. When one of the waves is significantly larger scale, such as an inertial wave, linear theory may be used to assess the interaction between it and a smaller scale wave. The result is three basic types of interactions:

1\textsuperscript{This work has been supported by the EPSRC and industrial partners in the Innovation in Inkjet Technology project.}
2\textsuperscript{Member of the APS}
3\textsuperscript{Member of the Institute of Physics (United Kingdom)}
4\textsuperscript{3This work has been partially supported by NSF HRD-0924212.}
L1.00015 Numerical study of internal waves due to time-dependent shear in the ocean, LEONARDO LATORRE, JULIE VANDERHOFF, Brigham Young University — The ocean and atmosphere have a particular characteristic that sustains propagation of internal gravity waves called a stable stratification. Internal waves are generated, with wavelengths which can vary from a few meters to kilometers. These waves propagate through the ocean and atmosphere exchanging energy and momentum as they interact with other fluid phenomena and break, which in turn affects circulation, heat transport, nutrient distribution and biological activity in the oceans and the atmosphere. However large scale circulation models lack the appropriate resolution to detect these motions, hence it is necessary to accurately parameterize internal wave breaking in order to establish a better relationship between wave energy dissipation and its effects on oceanic and atmospheric circulation patterns. In this research internal waves interact with a time dependent background in the form of a near-inertial wave, which are common in the ocean. Using a two dimensional, fully non-linear Navier-Stokes equation solver and ray theory, estimates of wave breaking parameters which predict breaking at the same location in both of these models are accomplished. A statistical analysis of waves observed during the Hawaiian Ocean Mixing Experiment will provide an estimate of the percentage of waves expected to break during propagation through an inertial wave.

L1.00016 On influence of microstructure on granular impact, XIAONI FANG, LOU KONDIC, NJIT, ROBERT BEHRINGER, Duke University, WOLFGANG LOSERT, University of Maryland, COREY O’HERN, Yale University — We use discrete element simulations to explore interaction of an intruder with a dense granular matter. Granular particles are modeled as soft, inelastic, frictional disks in two spatial dimensions, and the intruder is considered to be much larger than particle size. In this presentation we will concentrate in particular on the influence of granular microstructure on the impact, including the influence of system size, preparation, and material properties. The results will be compared to the existing ones, and new experiments will be proposed.

L1.00017 Nonlinear stability of granular shear flow: shear banding, PRIYANKA SHUKLA, PhD Student, MEHEBOOB ALAM, Prof. — We show that a Landau-type 'order-parameter' equation describes the onset of shear-band formation in granular plane Couette flow wherein the flow undergoes an ordering transition into alternate layers of dense and dilute regions of low and high shear rates, respectively, parallel to the flow-direction. Even though the linear theory predicts the stability of the homogeneous shear solution in dilute flows, our analytical bifurcation theory suggests that there is a sub-critical finite-amplitude instability that is likely to lead to shear-band formation in dilute flows which is in agreement with previous numerical simulations.

L1.00018 Testing the continuum $\mu(I)$ rheology for 2D granular flows on avalanches and collapse of columns, PIERRE-YVES LAGRÈE, LYDIE STARON, CNRS UPMC UJRA, STÉPHANE POPINET, NIWA — There is a large amount of experimental work dealing with dry granular flows (such as sand, glass beads, small rocks...) supporting the so called $\mu(I)$ rheology. This rheology states that the ratio of the tangential to the normal constraints behaves as a Coulomb like friction depending on the inertial number (this number is the product of the grain size by the shear of the velocity divided by the square root of pressure divided by the grain density). Hence, we propose the implementation of this non newtonian rheology in a Navier Stokes Solver (the Gerris Flow Solver uses a finite-volume approach with the Volume-Of-Fluid (VOF) method to describe variable-density two-phase flows). First we apply it on a steady infinite bi dimensional avalanching granular flow over a constant slope covered by a passive light fluid (it allows for a zero pressure boundary condition at the surface, bypassing an up to now difficulty which was to impose this condition on a unknown moving boundary). The classical analytical solution, known as Bagnold solution, is recovered numerically. Then the rheology is tested on the collapse of granular columns and quantitative comparisons with numerical simulations from Contact Dynamics are done.

L1.00019 Continuum Simulation of Impact into Granular Beds, ERIC WILKINSON, JON BOUGIE, Physics Department, Loyola University Chicago — We investigate the dynamics of objects impacting into a granular medium using continuum simulations. Although a static bed with long-lasting contact between grains exhibits a solid-like configuration, the bed may become locally fluidized near an impact by an external object. Studies of shock propagation through granular beds suggest grains may flow freely near the impact site, yielding behaviors that could be analyzed using a granular hydrodynamics approach. We test the ability of a set of proposed granular hydrodynamics equations to describe the dynamics of a granular bed following impact using a numerical simulation. This system provides a test case to study the applicability and limitations of a hydrodynamics approach for modeling granular systems with coexisting static and fluidized states. Additionally, this system could provide a basic model to develop a better understanding of a range of phenomena such as meteor impact, biological locomotion over sand, and the performance of protective materials.

L1.00020 Hydrodynamic Simulations of Density Inversion in Granular Layers, VERONICA POLICH, JON BOUGIE, Physics Department, Loyola University Chicago, JENNIFER KREFT PEARCE, Department of Chemistry, University of Texas at Tyler — We model density inversion in vertically shaken granular layers using a proposed set of three-dimensional, time-dependent granular hydrodynamics equations. For a range of shaking amplitudes and frequencies, we numerically solve time-dependent equations derived to Navier-Stokes order for mono-disperse, frictionless, nearly elastic particles. For shaking at high frequency and accelerational amplitude, these simulations exhibit steady state behavior in which a high density layer is supported by a lower density granular gas. At lower shaking frequencies and accelerational amplitudes, density profiles display time-dependence in which density inversion is not maintained throughout the entire cycle. Results from these simulations are directly compared to molecular dynamics simulations to test the ability of these continuum simulations to accurately model both the time-dependent and steady-state phenomena found in experiments and molecular dynamics simulations.

L1.00021 Continuum Simulation of Impact into Granular Beds, ERIC WILKINSON, JON BOUGIE, Physics Department, Loyola University Chicago — We investigate the dynamics of objects impacting into a granular medium using continuum simulations. Although a static bed with long-lasting contact between grains exhibits a solid-like configuration, the bed may become locally fluidized near an impact by an external object. Studies of shock propagation through granular beds suggest grains may flow freely near the impact site, yielding behaviors that could be analyzed using a granular hydrodynamics approach. We test the ability of a set of proposed granular hydrodynamics equations to describe the dynamics of a granular bed following impact using a numerical simulation. This system provides a test case to study the applicability and limitations of hydrodynamic approach for modeling granular systems with coexisting static and fluidized states. Additionally, this system could provide a basic model to develop a better understanding of a range of phenomena such as meteor impact, biological locomotion over sand, and the performance of protective materials.

1This research was supported by an award from Research Corporation for Science Advancement.

2This research was supported by an award from Research Corporation for Science Advancement.
Sato’s experiment. Interesting is it that the evolution of streamwise minus modes has a similarity to the symmetric eigenfunction as they develop into saturation. Developing, in the transitional to fully nonlinear regime, at rather bigger growth rates than the dominant spanwise modes. By examining the modal eigenfunction the anti-symmetric modes, grow exponentially in the linear regime and saturated after the nonlinear interactions. However, there are streamwise minus modes with experiment made by Sato in terms of linear growth rate and its eigenfunctions as well. In general the dominant spanwise modes (primary modes), usually excited by the random-broadband white noise in a sense of a natural experiment, will be further discussed in terms of jet instabilities from the beginning of noise range. Besides the validation of numerical simulations, the proposed decomposition can be used to reconstruct the periodic flow in order to minimize the series solution of the Hopf bifurcation and comparisons between the experimental results and FEM numerical simulations show good agreement in the considered dimensionless temperature, Nusselt number, and flow fields are among the presented results.

L1.00022 RNS computations of the boundary layer flow over grooved plate . JUAN MARTIN, CARLOS MARTEL, ETSI Aeronauticos, Universidad Politecnica de Madrid — We use the Reduced Navier-Stokes equations (RNS) to compute the evolution of the 3d boundary layer flow over a plate with small depth streamwise grooves carved in it. The RNS are derived from the Navier-Stokes equations for flows with large Re number with one slow scale and two short scales. The resulting RNS are nonlinear and fully parabolic equations. In this work we comment the details of the numerical integration of the RNS, where we use a conformal mapping to include the effect of the grooved bottom. We present the resulting flow structures due to the geometry of the problem, also a parametric study to evaluate the effect of the spanwise wave number over the flow configuration. The RNS computations are much more less CPU costly than full 3D DNS, and does not exhibit the numerical instabilities present in previous PSE calculations.

L1.00023 Direct route to turbulence in a rotating boundary layer , BERTRAND VIAUD, CREA-French Air Force, ERIC SERRE, M2P-CNRS, JEAN-MARC CHOMAZ, Ladhyx-CNRS — The transition to turbulence in a rotating boundary layer is analysed via DNS in an annular cavity made of two parallel co-rotating disks of finite radial extent, fed by a forced inflow at the hub. A former investigation [Viaud et al. JFM 2008] has established the existence of a primary subcritical bifurcation to nonrotating global mode with angular phase velocity and radial envelop coherent with the so-called elephant mode theory. When the Reynolds number based on the forced throughflow is increased above a threshold value for the existence of the nonlinear global mode, a large amplitude impulsive perturbation gives rise to a self-sustained saturated wave which is itself globally unstable. A second front appears in the lee of the primary where small-scale instability develops with characteristics indicating a Floquet mode of zero azimuthal wavenumber. This secondary instability leads to a very disorganized state, defining transition to turbulence. This transition, linked to the secondary instability of a global mode, confirms for the first time on a real flow the possibility of a direct transition to turbulence through an elephant cascade, a scenario up to now only observed in the Ginzburg–Landau model. Further work investigates the sensitivity of this scenario to environmental parameters, namely the streamwise extent of the flow, the incoming noise level, or the amplitude of the initial perturbation.

L1.00024 Coating of a cylindrical fibre: Instability and drop formation . ALEJANDRO G. GONZÁLEZ, JAVIER A. DIEZ, ROBERTO GRATTONÓ, Instituto de Física Arroyo Seco, Universidad Nacional del Centro de la Provincia de Buenos Aires, DIEGO CAMPANA, FERNANDO SAITA, Intec-Conicet, Universidad Nacional del Litoral, Guemes 3450, Santa Fe, Argentina — The instability of a liquid layer coating a thin cylindrical wire is studied experimentally and numerically with negligible gravity effects. The initial uniform film is obtained as the residual of a sliding drop, and the thickness measurements are performed with an anamorphic optical system. A primary mode grows in the early stages of the instability, and its wavelength \( \lambda_1 \) is not always in agreement with that predicted by the linear theory, \( \lambda_{10} \). In later stages, a secondary mode appears, whose wavelength is half that of the primary mode. The behavior of the secondary mode allows us to classify the experiments into two cases, depending on whether it is linearly stable (case I) or unstable (case II). In case I, the amplitude of the secondary mode remains small compared with that of the primary one, while in case II both amplitudes may become very similar at the end. Thus, the distance between the final drops may be quite different from that seen between initial protuberances. The analysis of the experiments allows us to define a simple criterion based on the comparison between \( \lambda_1 \) and \( \lambda_{10} \) (see Journal of Fluid Mechanics 651, 117 (2010)).

L1.00025 Modeling spreading of liquid crystal drops , LINDA CUMMINGS, TE-SHENG LIN, LOU KONDIC, New Jersey Institute of Technology — A series of experiments involving spreading of nematic liquid crystal drops on solid substrate have uncovered a surprisingly rich variety of behavior. The drops can either be arrested in their spreading, spread globally, or destabilize with or without spreading. We propose a relatively simple model which includes elastic contribution to the free energy as well as the finite anchoring energy due to the preferred orientation of the director field at the liquid/gas and liquid/solid interfaces. We find that the main features of the experiments, including spreading and instability regimes, can be in qualitative manner described by the proposed model.

L1.00026 A low-speed corona jet for internal spot cooling of tubes , MAJID MOLKI, REZA BAGHAEI LAKHEH, Southern Illinois University Edwardsville — A high electric potential applied to a wire electrode at the centerline of a circular tube may cause gas ionization and corona discharge, leading to formation of secondary flows and a corona jet within the tube. A computational model was implemented to show that the corona jet appears only if the electrode is slightly off-center with respect to the tube. The computations indicate that the jet is oriented in the direction of electrode-tube offset, and it may be suitable for target-cooling of thermal components mounted on the inner surface of the tube. Because the direction of the corona jet is adjusted by the orientation of the electrode, this arrangement may be used to focus the jet on specific areas for a more efficient and effective cooling of electronic components. In this study, the effectiveness of the aforementioned technique is investigated by reporting the local Nusselt number and rate of heat transfer. In addition, the corona-enhanced cooling is compared with the less efficient buoyancy-driven heat transfer in circular tubes. Variations of local dimensionless temperature, Nusselt number, and flow fields are among the presented results.

L1.00027 The Enhancement of Streamwise Minus Modes during Evolution of a Subsonic Compressible Jet Flow , HO-SHUENN HUANG — Direct numerical simulations in compressible jet flows within confined walls are studied. The results, excited by the random-broadband white noise in a sense of a natural experiment, will be further discussed in terms of jet instabilities from the beginning of noise level to the linear regime and transition to turbulence. An experimental study made of transition of a two-dimensional jet by Sato (1960) was chosen to run a similar direct numerical simulation in a temporal case and compared the results accordingly. Results of direct numerical simulations show a good agreement with experiment made by Sato in terms of linear growth rate and its eigenfunctions as well. General the dominant spanwise modes (primary modes), usually the anti-symmetric modes, grow exponentially in the linear regime and saturated after the nonlinear interactions. However, there are streamwise minus modes developing, in the transitional to fully nonlinear regime, at rather bigger growth rates than the dominant spanwise modes. By examining the modal eigenfunction of the root-mean-square fluctuating velocity in downstream direction, the dominant spanwise mode behaves an anti-symmetric eigenfunction as measured by Sato’s experiment. Interesting is it that the evolution of streamwise minus modes has a similarity to the symmetric eigenfunction as they develop into saturation.

L1.00028 Fourier decomposition of periodic flow using DPIV . IVAN KORKISCHKO, IAGO C. BARBEIRO, JULIO R. MENEGHINI, J.A.P. ARANHA, University of Sao Paulo — Digital particle image velocimetry (DPIV) is employed to measure the velocity fields of the flow past a circular cylinder in the 50<Re<340 range and a Fourier series based on the Strouhal period is employed to decompose the periodic time series up to the third harmonic. The series converges with decreasing coefficients and the final residual related to the DPIV initial data is very low. The first harmonic illustrates the von Kármán vortex street and its characteristic antisymmetric pattern with respect to the streamwise centerline. The second one is symmetric being responsible for the drag force main oscillations while the third is again antisymmetric. The harmonics respect the hierarchy suggested by the asymptotic series solution of the Hopf bifurcation and comparisons between the experimental results and FEM numerical simulations show good agreement in the considered Re range. Besides the validation of numerical simulations, the proposed decomposition can be used to reconstruct the periodic flow in order to minimize the effects of gappy data and measurement uncertainties. And for this case it is also very effective to decompose the flow in its most coherent structures.

Authors have grants from FINEP-CTPetro, FAPESP and Petrobras.
Penetration process and instabilities arise on a liquid jet impinged to a liquid flowing in a channel. KAORU HATTORI, Undergraduate, Department of Mechanical Engineering, Faculty Science & Technology, Tokyo University of Science, ICHIRO UEINO, Department of Mechanical Engineering, Faculty Science & Technology, Tokyo University of Science — We conduct a series of experiments with a special interest on a penetration process and instabilities arisen on a liquid jet impinged to a liquid of the same kind flowing in a channel. The impinged jet penetrates into the flowing bath accompanying with entrainment of the ambient immiscible gas, which results in the impinged jet wrapped by the entrained gas as a “sheath.” This sheath formation enables the impinged jet to survive in the fluid in the channel without coalescing until the entrained-air sheath breaks down. Occasionally a “cap” of the entrained air is formed at the tip of the penetrated jet, and the jet elongates like a long balloon. Dynamic behaviors of the penetrated jet and the departure of the bubble of warring gas at the tip of the collapsing jet observed by use of a high-speed camera are discussed.

Rayleigh Taylor Instability in the presence of Non-uniform Flow1, S. SEN, Lancaster University, UK — We study the Rayleigh–Taylor Instability in the presence of an equilibrium flow which varies across the cross section. It is found when the flow varies linearly with the radial coordinate (flow shear) the growth rate of the instability increases whereas for the quadratic variation with the radial coordinate (flow curvature) the mode is stabilized. This might have important implications in inertial confinement fusion where the stabilization of Rayleigh Taylor mode is one of the biggest obstacles for energy generation by fusion. This result will also have important implications in identifying the origin of various space fluctuations.

1Funding by European Commission (FP7) is acknowledged.

Richtmyer-Meshkov Instability: Effects of Mach Number and Initial Conditions on Turbulent Mixing, GAVIN FRIEDMAN, EXTREME FLUIDS TEAM — Effects of Mach number and initial conditions are studied on a thin Air-SF6-Air interface impulsively accelerated by a planar shock wave (Mach 1.2-1.8). A membraneless interface is formed using a nozzle design to create a stable gas curtain. Using particle image velocimetry and planar laser induced fluorescence, velocity and density fields are captured simultaneously to characterize the initial condition and the growth of the instability. To quantify and characterize the turbulent mixing, the evolving structure is reshocked at various times by varying the location of a moveable end wall. Turbulence statistics are compared between a single mode varicose curtain, and a multi-mode curtain. The results are compared with ongoing 3-D numerical simulations of the gas curtain experiment.
L1.00038 Molecular dynamics simulations of Janus nanoparticle assembly at interfaces, WEIKANG CHEN, JOEL KOPLIK, ILONA KRETZSCHMAR, CCNY — We study the formation of clusters of nano-sized Janus particles (having surface regions with different interactions) at a liquid-vapor interface using molecular dynamics simulations. The individual particles are modeled as rigid spherical sections of an atomic lattice, with short ranged atom-atom interactions chosen to selectively attract subregions of the particle interface, and with a solid-liquid interaction favoring a 90 degree contact angle at the liquid interface. The simulations automatically incorporate the competition between Brownian motion, fluid convection and molecular attraction, as well as evaporation of the liquid if desired. We study the distribution and shape of the clusters found in equilibrium, and the structures resulting when the solvent evaporates.

L1.00039 Atomistic simulation of water transport through graphene membrane, MYUNG EUN SUK, NARAYANA ALURU, University of Illinois — Graphene monolayer can be considered as the thinnest membrane reported so far as its thickness is only one carbon atom diameter. In this study, water transport through porous graphene membrane is investigated using molecular dynamics simulations. Water flux through graphene is compared to the water flux through thin (less than 10 nm in thickness/length) carbon nanotube (CNT) membranes at various diameters. For small diameter, where single-file structure is observed, water flux is lower through the graphene membrane compared to that of the CNT membrane. On the other hand, for larger diameter pores, where the single-file structure is no longer observed, water flux is higher through the graphene membrane, compared to that of the CNT membrane. We explain the results using hydrogen bonding dynamics, pressure distribution and potential of mean force.

L1.00040 Experimental Evaluation of Simplified Theoretical Models for Fluid Mud Gravity Current Propagation, FIRAT TESTIK, MIJANUR CHOWDHURY, Civil Engineering Department, Clemson University — The propagation dynamics of fluid mud gravity currents were studied experimentally and theoretically. The experimental currents propagate under slumping, self-similar and viscous phases. The transition times from slumping to self-similar and from self-similar to viscous phases are parameterized. Predictive capabilities of the three existing theoretical modeling approaches (force-balance, box and shallow water) were evaluated based on our experimental observations. For the slumping and self-similar phases, both the force-balance and box model solutions showed a better predictive capability than the one-layer shallow water model solution. Having non-Newtonian rheology, the propagation dynamics of fluid mud gravity currents in the viscous phase vastly differ from the Newtonian currents. A force-balance expression for the slumping of a two-layer fluid mud gravity current was derived. The predictions of this force-balance expression and a recent viscous shallow water model solution are observed to be in good agreement with the experimental data. The results of this study are expected to be useful in predicting the spreading of fluid mud gravity flows that occur in different natural and industrial situations.

L1.00041 Permeability prediction of isotropic fibrous porous media, SONIA WOUDBERG, J. PRIEUR DU PLESSIS, Stellenbosch University — Fibrous porous media find application in several industrial engineering disciplines including filtration processes and fuel cells. In the present study a geometric pore-scale model is introduced and used to predict the permeability of isotropic fibrous porous media. The model is based on a unit cell approach in which the fibres of the actual porous medium are modelled based on rectangular geometry. At first the model is used to predict the permeability of cross-flow through an array of unidirectional fibres. The permeability is expressed as a function of the solid volume fraction and a pore-scale linear dimension. In addition a three-dimensional isotropic model is proposed by performing a weighted average on the model for cross-flow and a model from the literature for flow parallel to the fibre axes. The resulting model is compared to a comprehensive collection of experimental data from numerous authors, based on various types of fibrous porous media, including that of entangled polymer networks. The Kozeny “constant” is calculated for different solid volume fractions and it is illustrated that the pore-scale model introduced conserves the constancy of the Kozeny constant.

L1.00042 ABSTRACT WITHDRAWN

L1.00043 Evolution of droplets of perfectly wetting liquid under the influence of thermocapillary forces, SHOMEEK MUKHOPADHYAY, Chemistry, Columbia University, NEBOJSA MURISIC, Mathematics, UCLA, ROBERT P. BEHRINGER, Physics, Duke University, LOU KONDIC, Mathematical Sciences, New Jersey Institute of Technology — We consider evolution of sessile droplets of a nonvolatile perfectly wetting liquid on differentially heated solid substrates. The heating induces thermocapillary Marangoni forces which affect the contact line dynamics. In our experiments, we witness the opposing action of the thermocapillary Marangoni effect and capillary spreading. We record an interesting feature which develops during this phase — while the bulk of the drop mass recedes toward the center, the contact line recedes at a much slower rate, leaving a stretched layer of liquid between the main body of the drop and the contact line. We find that this layer of liquid thins as evolution of the drop proceeds and that the thinning is more pronounced when the imposed temperature gradient in the contact line region is larger. Our theoretical model, based on the lubrication approximation and incorporating the Marangoni effect, recovers the main features observed in the experiments. The model also indicates a strong dependence of the drop shape on the imposed temperature gradient, and, for a particular class of temperature profiles, it predicts formation of a ridge between the thin liquid layer and the main body of the drop, which is still to be observed in experiments.

L1.00044 Characterization of hydrophobic and hydrophilic coatings as deicing and anti-icing, AKIHITO AOKI, Kagokuin University, KATSUAKI MORITA, University of Tokyo, AKIHISA KONNO, Kagokuin University, HIROTAKA SAKAUE, JAXA — Anti-icing is necessary in various fields, such as aeronautics, roads, power lines, ships, and architectures. Deicing fluids, and sometimes hot water, work to prevent from icing. Due to environmental issue, deicing fluids are not always welcome to use. We study hydrophobic and hydrophilic coatings for anti-icing. By coating these to a target surface, it prevents icing without damaging the environment. We present a characterization method of hydrophobic and hydrophilic coatings for deicing and anti-icing. We provide a temperature-control room to create an icing condition, such as -10 to 0 degrees C. Under the controlled room, the contact angle measurement as well as the force measurement is employed. Total 15 coatings are characterized. Based on the tests of all coatings, we propose a combined coating from some characterized ones.

L1.00045 Lagrangian particle tracking in strained turbulence, DIMITRY IVANOV, RÖBERT GRÖNVIST, Reykjavik University, CHUNG-MIN LEE, California State University at Long Beach, ÅRMANN GYLFASON, Reykjavik University — We present initial results from experimental investigations of axisymmetrically strained turbulent flow. Our focus is on the influence of the straining on the motions of passive and inertial particles. The results are compared with existing numerical and experimental data, and we seek to emphasize the effects of the strain geometry and strain rate on the particle behaviour. Eulerian, PIV, flow field results are also presented. We furthermore present a new approach for the analysis and processing of particle tracks and discuss our experimental errors in detail.

L1.00046 Viscous Flow past a Semi-Infinite Plate, LING XU, MONIKA NITSCH, University of New Mexico — Numerical investigations are made to simulate two dimensional viscous fluid flows past a semi-infinite plate at early short time. Details of the flow structures are visualized using a finite difference method. The shedding of the vortex sheet from the plate tip is studied.
L1.00047 On the potential for transport via internal tides1. GONCALO GIL, OLIVER FRINGER, Stanford University — Non-linear effects associated with internal waves lead to advection of fluid particles along with suspended mass such as sediment, nutrients, larvae, as well as contaminants. These factors contribute to the development of benthic communities, the geological shaping of the continental slope and, in some situations, play a role in the transport and fate of contaminants. We compute particle trajectories and resulting Stokes velocity profiles using a Navier-Stokes code with a Lagrangian particle tracking model. Results are compared to linear theory and a semi-nonlinear formulation using a uniform stratification and stratification typically found at Huntington Beach, CA where there is recurring bacteriological contamination.

L1.00048 On the potential for transport due to internal tides in the coastal ocean1. GONCALO TRIGO CEBRITA GIL, OLIVER FRINGER, Environmental Fluid Mechanics Laboratory Dept. of Civil and Environmental Engineering Stanford University, Stanford, CA 94305 — Non-linear effects associated with internal waves lead to advection of fluid particles along with suspended mass such as sediment, nutrients, larvae, as well as contaminants. These factors contribute to the development of benthic communities, the geological shaping of the continental slope and, in some situations, play a role in the transport and fate of contaminants. We compute particle trajectories and resulting Stokes velocity profiles using a Navier-Stokes code with a Lagrangian particle tracking model, both are second-order accurate in time and in space. Results are compared to linear theory and a semi-nonlinear formulation using a uniform stratification and stratification typically found at Huntington Beach, CA where there is recurring bacteriological contamination.

L1.00049 Internal wave attractors in stratified fluids, robustness to perturbations. LEO MAAS, NIOZ-Netherlands Institute for Sea Research, JEROEN HAZEWINKEL, Scripps - UCSD, CHRYSANTHI TSIMITRI, EAWAG - Switzerland, STUART DALZIEL, GK Batchelor lab DAMTP Cambridge UK — Previously, internal wave attractors have been studied in the laboratory in idealized situations. Here, we present a series of experiments in which these conditions are modified. Modifications are made by varying the forcing frequency, by using a non-uniform stratification, by introducing a periodic perturbation to the oceanic domain and by using a parabolic domain. All these new experiments reveal the persistence of internal wave attractors that remain reasonably well predictable by means of ray tracing. We conclude that the possibility of wave attractors has to be addressed whenever internal waves are found in stratified fluids.

Monday, November 22, 2010 3:35PM - 5:32PM Session LA Turbulent Mixing III Long Beach Convention Center 101A

3:35PM LA.00001 Regularization mechanism of Rayleigh-Taylor turbulent mixing1, SNEZHANA I. ABRARZHI, University of Chicago — Turbulent mixing induced by Rayleigh-Taylor instability plays an important role in a variety of natural and artificial phenomena spanning astrophysical and low to high energy density regimes. We apply group theory to analyze symmetries, invariants, scaling and spectra of turbulent mixing induced by the Rayleigh-Taylor instability. The properties of this statistically unsteady, anisotropic, and inhomogeneous turbulent process are found to depart from the canonical Kolmogorov scenario. Time- and scale-invariance of the rate of momentum loss leads to non-dissipative momentum transfer between the scales, to 1/2 and 3/2 power-law scale dependencies of the velocity and Reynolds number respectively, and to spectra distinct from Kolmogorov. Turbulent mixing exhibits more order compared to isotropic turbulence and its viscous and dissipation scales are set by the flow acceleration. To trigger relaminarization of RT mixing, few mechanisms are proposed, including highly coherent initial conditions and flow acceleration by high favorable pressure gradient.

3:48PM LA.00002 Vortex interactions and mixing in large Atwood number two-dimensional turbulence, LAURENT JOLY, Universite de Toulouse, ISAE, JEAN NOEL REINAUD, University of St Andrews, JEROME FONTANE, Universite de Toulouse, ISAE — We perform direct numerical simulations of the relaxation of variable-density two-dimensional turbulence. An initial collection of vortices is evolved in time together with an initially bimodal density field corresponding to an Atwood number, LAURENT JOLY, Universite de Toulouse, ISAE, JEAN NOEL REINAUD, University of St Andrews, JEROME FONTANE, Universite de Toulouse, ISAE — Non-linear effects associated with internal waves lead to advection of fluid particles along with suspended mass such as sediment, nutrients, larvae, as well as contaminants. These factors contribute to the development of benthic communities, the geological shaping of the continental slope and, in some situations, play a role in the transport and fate of contaminants. We compute particle trajectories and resulting Stokes velocity profiles using a Navier-Stokes code with a Lagrangian particle tracking model, both are second-order accurate in time and in space. Results are compared to linear theory and a semi-nonlinear formulation using a uniform stratification and stratification typically found at Huntington Beach, CA where there is recurring bacteriological contamination.

The work is supported by the US National Science Foundation and by the US Department of Energy.

4:01PM LA.00003 Transport, dispersion and mixing in quasi-two-dimensional steady jets, J.R. LANDEL, BPI & DAMTP, U. of Cambridge, C.P. CAULFIELD, (BPI & DAMTP), ANDREW W. WOODS, (BPI) — The study of turbulent jets in relatively enclosed geometries is relevant to many chemical engineering processes. Predicting the concentration of chemical reactants in space and time requires a good understanding of the jet dynamics. We consider experimentally and theoretically the behaviour of liquid jets in a quasi-Hele-Shaw cell, where the jets are constrained in a narrow gap whose width is two orders of magnitude smaller than the other two flow dimensions. Classical theoretical models for plane jets are in excellent agreement with time-averaged experimental results obtained using both dye jets and PIV techniques. Detailed examination of instantaneous structures of the flow reveals a high-speed sinuous core at the centre of the jet and large vortical structures on each side, which we analyse quantitatively using a variety of techniques. These structures have a large impact on the mixing and dispersion properties of the jet. We use a virtual-particle-tracking technique to assess and understand this effect. Comparisons between the instantaneous and the time-averaged velocity field show the importance of the inherently time-dependent vortical structures in the mixing and stretching of the fluid, substantially modifying the mixing and (vertical) dispersion within the jet.

4:14PM LA.00004 Shape Dynamics of Lagrangian Clusters in Two-dimensional Flow. ALEXANDRE DE CHAUMONT QUITRY, Yale University, DOUGLAS KELLEY, SOPHIA MERRIFIELD, NICHOLAS OUELLETTE — In an effort to understand the dispersion of passive scalars in two-dimensional flows, we investigate the shape evolution of three-particle clusters. We compute the trajectories of virtual Lagrangian points in an electromagnetically driven experimental flow. While our working flow is not turbulent, we observe the same stationary isotropic limit previously observed in turbulent flows. Further, we find that at different scales the triangles adopt preferred statistical shape distributions insensitive to their initial configuration. Our results thus emphasize the role of scale-dependent Lagrangian flow structures in the mixing process. This work is supported by the National Science Foundation.
4:27PM LA.00005 Influence of flow topology on Lagrangian statistics in forced 2-D turbulence

Benjamin Kadock, M2P2/ECM/USD, Diego Del-Castillo-Negrete, Oak Ridge National Laboratory, Wouter Bos, LMFA/Ecole Centrale Lyon, Kai Schneider, M2P2/Université de Provence — Conditional Lagrangian statistics of forced two-dimensional turbulence in unbounded and bounded domains are studied by means of direct numerical simulation. The instantaneous flow domain is decomposed into either vorticity or strain-dominated regions and a quiescent background region using the Okubo-Weiss criterion. The probability distribution function (PDF) of the residence time of the particles exhibits an exponential behavior in the background, while in the vorticity and strain-dominated regions self-similar algebraic tails are found. For Lagrangian acceleration it is shown that both the vorticity and strain region are responsible for the heavy tails. Finally, the conditional PDFs of the curvature are found to be independent of the different flow domains yielding algebraic tails with slope close to -2, characteristic for an inverse chi-square distribution.

4:40PM LA.00006 Experimental Study of Mixing dynamics in Stratified Jet

Duò Xu, Jun Chen, Purdue University — Stratification due to density difference or temperature difference modifies flow structures significantly. In order to characterize the mixing process in stratified flows, momentum and scalar flow terms are to be analyzed. In this study, Particle Image Velocimetry (PIV) and planar laser induced fluorescence (PLIF) are applied to simultaneously measure velocity and density fields generated by a horizontal stratified turbulent jet. The effects of stable stratification and unstable stratification are examined. Flow dynamics at two characteristic Richardson numbers is analyzed by examining the development of flow statistics. The dataset is also applied to test different mixing models.

4:53PM LA.00007 Variational Theory of Hyperbolic Lagrangian Coherent Structures

George Haller, McGill University — We describe a mathematical theory that clarifies the relationship between Lagrangian Coherent Structures (LCS) in unsteady fluid flows and invariants of the Cauchy-Green strain tensor field. Motivated by physical observations of tracer patterns, we define hyperbolic LCS as material surfaces that extremize an appropriate finite-time normal repulsion or attraction measure. Solving this variational problem leads to computable sufficient and necessary criteria for LCS. We also discuss constrained LCS problems, as well as the robustness of LCS under numerical errors and data imperfections. In several examples, we show how these results resolve earlier inconsistencies in the theory of LCS.

5:06PM LA.00008 Separating stretching from folding in fluid mixing

Douglas H. Kelley, Nicholas T. Ouellette, Yale University — Efficient large-scale mixing depends on stretching and folding — together they expand the periphery of material volumes, allowing diffusion to mix at small scales. Yet stretching and folding are difficult to decouple in real flows with complex spatiotemporal structure. We distinguish the two processes mathematically and study them separately in a laboratory flow. Our experimental apparatus is a quasi-two-dimensional electromagnetically driven stratified solution with lateral dimensions 90 cm × 90 cm. Optically tracking ∼30 000 particles per frame with a high-speed camera, we reconstruct the velocity field and express fluid deformations as the unique sum of an affine component (primarily stretching) and a non-affine component (primarily folding). At short times stretching dominates, but once fluid elements have elongated, folding becomes suddenly stronger and dominates thereafter. The relative strength of the two processes also varies strongly in space. This work is supported by the National Science Foundation.

5:19PM LA.00009 Experimental evidence of a phase transition in a closed turbulent flow

François Daviaud, Pierre-Philippe Cortet, Arnaud Chiffaudel, Berengère Dubrulle, CEA, IRAMIS, SPEC, CNRS URA 2464, Groupe Instabilities & Turbulence, 91911 Gif-sur-Yvette, France — Using stereoscopic particle image velocimetry, we experimentally study the susceptibility to symmetry breaking of a weakly turbulent Karman swirling flow for $Re = 150$ to $Re ≃ 10^4$. The susceptibility of the mean flow is shown to increase from 1 to 1.5 as transition to turbulence proceeds from the laminar flow to the highly turbulent flow. We report a divergence of this susceptibility at an intermediate Reynolds number $Re = Re_c ≃ 90 000$ which gives experimental evidence that such a highly space and time fluctuating system can undergo a “phase transition”. This transition is furthermore associated with a peak in the amplitude of fluctuations of the instantaneous flow symmetry corresponding to intermittencies between metastable states. These states break spontaneously the symmetry of the forcing while the very long time-averaged mean flow respects the forcing symmetry.

Monday, November 22, 2010 3:35PM - 5:45PM
Session LB Turbulent Boundary Layers VI
Long Beach Convention Center 101B

3:35PM LB.00001 Boundary layer flow over streamwise grooves

Carlos Martel, Juan Ángel Martín, Universidad Politécnica de Madrid, DENLIA TEAM — We use the Reduced Navier-Stokes (RNS) equations for the simulation of the nonlinear evolution of a zero pressure gradient boundary layer flow over a grooved bottom wall. The RNS formulation provides Reynolds independent solutions that are asymptotically exact in the limit $Re ≫ 1$. It requires much less computational effort than DNS and it is numerically more robust than nonlinear PSE. We present results for the different flow patterns that appear depending on the spanwise period of the grooves and their cross section profile. And we discuss the idea of using the grooves to induce transversal motion in the boundary layer in order to produce a stabilization effect similar to the one induced by the streaks in a flat plate boundary layer.

3:48PM LB.00002 Characteristics of Turbulent flow over Superhydrophobic Surfaces

Jennifer Franck, Charles Peguero, Brown University, Charles Heno, NUWC, Kenneth Breuer, Brown University — Recent research has suggested significant modification in the structure of turbulent flow of water over a superhydrophobic surface. The changes, which may include large reductions in skin friction, are due to the modification of the no-slip boundary condition at the liquid-solid interface. We present experimental and computational results from an ongoing exploration of this system. Experimental results include new measurements of laminar flow friction coefficients, as well as high-resolution PIV over a number of superhydrophobic geometries. To complement the experimental investigations, direct numerical simulations of turbulent channel flow are performed. The no-slip boundary layer is modified with Navier slip boundary conditions in the streamwise and spanwise flow directions. The effect of compliance at the air-water interface between microstructures is investigated numerically using a simple model to calculate out-of-plane wall deflections and allow for non-zero wall-normal velocities. Mean and fluctuating velocity statistics as well as flow structures are examined, and compared with the experimental measurements.

4:01PM LB.00003 Analysis of the Mean Momentum Balance in Polymer Drag Reduced Turbulent Boundary Layers

Christopher White, Matt Blake, Joe Klevicki, University of New Hampshire, Yves Dubief, University of Vermont — Mean momentum balances (MMB) in polymer drag reduced turbulent channel and zero-pressure gradient boundary layer flows are examined using experimental and numerical data available in the literature. For each data set, three flow cases are examined: Newtonian, low drag reduction (LDR), and high drag reduction (HDR). The Newtonian case is used as a baseline for comparison, while LDR and HDR flows are chosen since turbulent statistics trend differently between LDR and HDR flows. The results show that important qualitative features of the layer structure that exists for flows of a Newtonian fluid exist for flows of drag reducing polymer solutions. However, with increasing drag reduction: the stress gradient balance layer extends further from the wall, the Reynolds stress gradient contribution to the MMB decreases, and the polymer stress gradient contribution to the MMB increases. The latter finding demonstrates that polymers have a significant effect on the mean dynamics of HDR flows.

1 Funded by Grants from NSF and ONR.
2 Summer Research Assistant
4:14PM LB.00004 Air-Induced Drag Reduction at High Reynolds Numbers: Velocity and Void Fraction Profiles, BRIAN ELBING, SIMO MÄKIHARJU, ANDREW WIGGINS, DAVID DOWLING, MARC PERLIN, STEVEN CECCIO, University of Michigan — The injection of air into a turbulent boundary layer forming over a flat plate can reduce the skin friction. With sufficient aerodynamic forces an air layer can separate the solid surface from the flowfield, which can produce drag reduction in excess of 80%. Several large scale experiments have been conducted at the US Navy’s Large Cavitation Channel on a 12.9 m long flat plate model investigating bubble drag reduction (BDR), air layer drag reduction (ALDR) and the transition between BDR and ALDR. The most recent experiment acquired phase velocities and void fraction profiles at three downstream locations (3.6, 5.9 and 10.6 m downstream from the model leading edge) for a single flow speed (~0.4 m/s). The profiles were acquired with a combination of electrode point probes, time-of-flight sensors, Pitot tubes and an LDV system. Additional diagnostics included skin-friction sensors and flow-field image visualization. During this experiment the inlet flow was perturbed with vortex generators immediately upstream of the injection location to assess the robustness of the air layer. From these, and prior measurements, computational models can be refined to help assess the viability of ALDR for full-scale ship applications.

1Sponsored by the Office of Naval Research.

4:27PM LB.00005 Local suppression of turbulent noise by passively inducing relaminarization, RICHARD KIRKMAN, MEREDITH METZGER, University of Utah — Direct numerical simulations of turbulent channel flow were performed to study potential means of locally suppressing wall pressure noise by passively driving the flow towards relaminarization. The noise reduction is achieved by altering the surface geometry along a wall of the channel. Two separate geometries were investigated, namely a wedge-shaped protrusion and an inverted wedge-shaped depression. In both configurations, the wedge remains stationary and spans the width of the channel. The flow tends toward relaminarization due to local convective acceleration along the upslope of the wedge (in the case of the protrusion) and due to the gradual unstalled expansion along the downslope of the wedge (in the case of the depression). Simulations were performed at a Reynolds number based on friction velocity of 180. The no-slip condition along the surface of the protrusion/depression was enforced using an immersed boundary method. Profiles of turbulence statistics and wall-pressure intensity, as well as the wall-pressure spectra along the front face of the two different wedges are compared in relation to those of the undisturbed approach boundary layer.

2 Funded by the DARPA STTR program in collaboration with Progeny Systems Corp.

4:40PM LB.00006 The coherent structure of atmospheric surface layers, KAPIL CHAUHAN, NICK HUTCHINS, IVAN MARUSIC, JASON MONTY, The University of Melbourne — The structure of two-point correlation statistics in the atmospheric surface layer is studied from measurements on the western Utah salt flats at the SLTEST facility. Large-scale features in the stable, neutral and unstable surface layers that adhere to Monin-Obukhov similarity (~10 ≤ z/ζ ≤ 1) are observed and are examined through two-point correlation of streamwise fluctuations (zz). The changes in Rzz with changing z/ζ are quantified by evaluating the integral length scales in the streamwise and spanwise directions and the structure inclination angle. Both streamwise and spanwise integral length scales show consistent logarithmic trends that increase with decreasing stability. Further, the structure inclination angle in the wall-normal plane also shows a logarithmic increase with increasing z/ζ for the unstable surface layers. The changes in structure Rzz can be characterized by z/ζ making it feasible to incorporate the trends in near-wall models of LES of atmospheric flows under stable and unstable conditions.

4:53PM LB.00007 Very-Large-Scale Coherent Structures in the Wall Pressure Field Beneath a Supersonic Turbulent Boundary Layer, STEVEN BERESH, JOHN HENFLING, RUSSELL SPILLERS, BRIAN PRUETT, Sandia National Laboratories — Previous wind tunnel experiments up to Mach 3 have provided fluctuating wall-pressure spectra beneath a supersonic turbulent boundary layer, which essentially are flat at low frequency and do not exhibit the theorized ω2 dependence. The flat portion of the spectrum extends over two orders of magnitude and represents structures reaching at least 100 δ in scale, raising questions about their physical origin. The spatial coherence required over these long lengths may arise from very-large-scale structures that have been detected in turbulent boundary layers due to groupings of hairpin vortices. To address this hypothesis, data have been acquired from a dense spanwise array of fluctuating wall pressure sensors, then invoking Taylor’s Hypothesis and low-pass filtering the data allows the temporal signals to be converted into a spatial map of the wall pressure field. This reveals streaks of instantaneously correlated pressure fluctuations elongated in the streamwise direction and exhibiting spanwise alternation of positive and negative events that meander somewhat in tandem. As the low-pass filter cutoff is lowered, the fluctuating pressure magnitude of the coherent structures diminishes while their length increases.

5:06PM LB.00008 Vortex convection velocities in wall parallel planes of a turbulent boundary layer, JEFFREY A. LEHEW, MICHELE GUALA, BEVERLEY J. MCKEON, California Institute of Technology — The organization and convection velocity of vortices in wall parallel planes of a zero-pressure gradient turbulent boundary layer are investigated using time resolved digital particle image velocimetry (DPIV) at a moderate Reynolds number (Reτ = 470). Time resolved DPIV provides a means for tracking vertical structures in the flow giving their trajectories, velocities, and relation to other turbulent structures in the flow. Measurements are taken at three different wall normal locations (y/δ ≈ 0.07, 0.23, and 0.59) and comparisons of the vortex populations and convection velocities are made between the three planes. Vortical structures captured in these planes may be interpreted as signatures of hairpin-like structures which have been proposed to play a key role in turbulent boundary layer dynamics.

3Support from AFSOR under award # FA9550-09-1-0701 (program manager John Schmisser) is gratefully acknowledged.

5:19PM LB.00009 Analysis of vortex populations in turbulent boundary layers based on tomographic PIV, QI QAO, CECILIA ORTIZ-DUENAS, ELLEN LONGMIRE, AEROSPACE ENGINEERING AND MECHANICS, UNIVERSITY OF MINNESOTA TEAM — Vortex populations in the logarithmic region of turbulent boundary layers were investigated using results from tomographic PIV. The experiments were carried out in a water channel facility with δ = 125 mm and Reτ = 2500 (Reτ = 6200). Measurement volumes were about 90 x 80 x 9 mm3 (1650 x 1470 x 130 viscous units) spanning a wall-normal range from z+ = 150 to 280. Four 2K x 2K cameras were mounted above the channel and aimed at the measurement volume with tilt angle about 30 degrees to the wall normal direction. The magnification was 0.07 mm/pixel. Correlations were performed on 48 x 48 x 48 voxel volumes with 75% overlap yielding a vector space of 17 x 17 x 17 viscous units. Swirl strength and swirl direction were used to identify and characterize vortices in terms of orientation, circulation, size, and convection velocity. The results showed that swirl direction was a better indicator than vorticity of eddy orientation. Eddy circulation was found to increase approximately quadratically with eddy radius. The advantages and limitations of tomographic PIV vs. dual plane PIV will be discussed.

1Supported by NSF (CTS-0324898).
Quantitative evaluations of this method are performed to computer-generated images and actual PIV measurements. The behavior of this reflection is critical in determining the performance characteristics of the pump and understanding how to design practical valveless impedance pumps. In this study we investigate the pump performance and motion of the pump wall and working fluid in a 20mm x 10mm x 500 µm planar impedance pump. Elastic membranes impregnated with tracer particles were fabricated in-house with varying mechanical properties. For the first time, using the 3-dimensional DPIV technique known as defocusing digital particle image velocimetry (DDPIV), we were able to simultaneously track the motions of the elastic pump wall and the fluid within the pump. We study the partial pulse reflection at the end of the pump where the elastic membrane is coupled to the rigid flow loop. The behavior of this reflection is critical in determining the performance characteristics of the pump and understanding how to design practical impedance pump devices for microfluidic applications.

3:48PM LC.00002 Stereo-PIV study of turbulent flow downstream of a bend in a round pipe1

JUN SAKAKIBARA, AKIRA HASHIMOTO, Department of Engineering Mechanics and Energy, University of Tsukuba — We measured three-components of velocity vector distribution in cross sections of a fully developed turbulent pipe flow downstream of a 90-degree bend by means of stereo PIV. Reynolds number was $Re = 120,000$, and ratio of inner diameter $d$ of the pipe and radius of the centerline of the bend was 1.5. Temporal and spatial evolution of turbulent Dean type of vortices has been captured. Proper orthogonal decomposition (POD) was applied to the velocity field in cross-sections. At a streamwise distance $z = 2d$ downstream from the bend, the power of the 1st and 2nd mode structures identified at $z/d = 2$ were switched further downstream, and the structure of 2nd mode at $z/d = 2$ tends to be dominant. Reconstruction of the velocity vector field based on the 1st mode at $z/d = 2$ with mean velocity vectors gives flow pattern similar to that of swirl switching, while the 2nd mode at $z/d = 2$ is responsible to the rotation of the symmetry plane of the twin vortices.

1This work was supported by Tokyo Electric Power Company.

4:01PM LC.00003 Simultaneous DDPIV of the elastic wall and working fluid near the reflection sight of a valveless impedance pump1

JOHN MEIER, DEREK RINDERKNECHT, MORTEZA GHARIB, California Institute of Technology — Predicting the pressure and flow behavior in microscale impedance pumps is crucial for implementing the pump into microfluidic devices. Studies by Hickerson and Gharib (J. Fluid Mech. 2006) and Avrahami and Gharib (J. Fluid Mech. 2008) highlight the role of wave dynamics, reflections, and resonance in valveless impedance pumps. In this study we investigate the pump performance and motion of the pump wall and working fluid in a 20mm x 10mm x 500µm planar impedance pump. Elastic membranes impregnated with tracer particles were fabricated in-house with varying mechanical properties. For the first time, using the 3-dimensional DPIV technique known as defocusing digital particle image velocimetry (DDPIV), we were able to simultaneously track the motions of the elastic pump wall and the fluid within the pump. We study the partial pulse reflection at the end of the pump where the elastic membrane is coupled to the rigid flow loop. The behavior of this reflection is critical in determining the performance characteristics of the pump and understanding how to design practical impedance pump devices for microfluidic applications.

1Graduate student support graciously provided by the NDSEG

4:14PM LC.00004 PIV measurement of flow around an arbitrarily moving body1

YOUNG JIN JEON, HYUNG JIN SUNG, KAIST — PIV image processing methods for measuring flow velocities around an arbitrarily moving body are proposed. A contour-texture analysis based on user-defined textons is applied to determine the arbitrarily moving interface in the 2D PIV. After the interface tracking procedure is performed, the particle images near the interface are transformed into Cartesian coordinates that are related to the distance from the interface. This transformed image always has a straight interface, so the interrogation windows can easily be arranged at certain distances from the interface. Accurate measurements near the interface can then be achieved by applying the window deformation algorithm in concert with PIV/IG. For a tomographic 3D PIV, a volume reconstruction technique from four views is applied to obtain a three-dimensional shape of the interface. Particle motion analysis is made by the MTE MART algorithm. Quantitative evaluations of this method are performed to computer-generated images and actual PIV measurements.

1This work was supported by the Creative Research Initiatives of MEST/NRF.

4:27PM LC.00005 Scalable Tomographic PIV using a Reprojection Reconstruction Technique

RODERICK LA FOY, SAMUEL RABEN, PAVLOS VLACHOS, Virginia Tech — Tomographic PIV is becoming a common experimental tool in fluid dynamics, but current algebraic reconstruction algorithms can be prohibitively computationally expensive. To this end, a tomographic reconstruction algorithm was developed that is simple to implement, computationally efficient, and scalable to an arbitrary number of cameras. This method reconstructs volumes from each camera independently by projecting the images back onto the volume using the camera calibration data. The projections from each camera are then combined to form a full three dimensional intensity field. The fluid's velocity field may then be calculated using standard three dimensional PIV or PTV techniques. This algorithm was used to reconstruct intensity fields from both simulated and experimental particle fields.

4:40PM LC.00006 Reduction in the divergence of Tomographic P.I.V. results, using the method of Projection Onto Convex Sets

THOMAS CLARK, TIMOTHY NICKELS, University Of Cambridge — Tomographic PIV allows measurement of fully three-dimensional velocity fields within a region of fluid. Experimental error in the measurement process, combined with the differential nature of the divergence operator results in a large error in the divergence of the results. For incompressible flow, a technique is introduced to reduce the divergence of the output results using the method of Projection Onto Convex Sets (POCS). The “correction” is formulated using a Lagrangian multiplier and implemented through solution of the Poisson equation in a cuboid domain. In this talk, the formulation of the “correction” and the boundary conditions used are presented along with experimental results. Limitations of the technique and possible applications are discussed.
4:53PM LC.00007 Measurement bias in intensity-based near-surface particle velocimetry due to tracer size variations, WEI WANG, Binghamton University, JEFFREY GUASTO, Massachusetts Institute of Technology, PETER HUANG, Binghamton University — Near-surface particle-based velocimetry using evanescent wave microscopy has gained popularity for experimental studies in biophysical
transport and nanofluidics. This technique is capable of measuring the 3D motion of small fluorescent tracer particles (10 nm to 10 micron) within a few hundred nanometers of a liquid-solid interface. Particle intensity is mapped to distance from the interface using the monotonic decay of the evanescent wave. In this work, we consider the measurement bias introduced by polydisperse tracer particle size on the measured spatial distribution of particles and their implications for nano-scale velocimetry. We present a general model to account for particle size variation and the associated interaction potentials between tracer particles and the solid interface (e.g. electrostatic and van der Waals forces) under typical experimental conditions.

5:06PM LC.00008 Synchrotron X-Ray Three Dimensional microPIV, ELIZABETH VOIGT, RODERICK LA FOY, KAMEL FEZZAA, WAH KEAT LEE, PAVLOS VLACHOS, Virginia Tech — A study was completed to validate the ability to use x-ray imaging combined with a tomographic algorithm to reconstruct a time resolved three dimensional flow field. In this study an approximately 1 mm cubed region was imaged using a synchrotron x-ray source. The x-ray beam was split in two, projected through the fluid interrogation volume, and focused onto a scintillating crystal. The scintillators were imaged using two high-speed cameras. A variety of flows were tested using 10 micron hollow glass spheres as tracer particles. The images from the cameras were combined using a tomographic algorithm and the velocity fields were calculated using three dimensional PIV and PTV methods.

5:19PM LC.00009 Development of 3D tomographic X-ray PIV technique1, SUNG YONG JUNG, SANG JOON LEE, Center for Biofluid and Biomimic Research, Department of Mechanical Engineering, POSTECH, Korea — An X-ray tomography particle image velocimetry (PIV) technique employing a medical X-ray tube as a light source was developed to measure three-dimensional velocity field information of various fluid flows. The PIV velocity field measurement technique has been used to extract velocity vectors of tracer particles seeded in a flow by tracing their displacements. The conventional PIV techniques using visible light are inappropriate to measure flows in opaque conduits. To overcome these limitations on special applications, the x-ray PIV technique has been developed. In the x-ray imaging technique, the volumetric information along the pathway of x-ray propagation was compressed on the projected image. Therefore, the X-ray computed tomography has been employed to reconstruct the three-dimensional structure of opaque materials using multiple X-ray images captured at several different angles. As a result, we could successfully reconstruct a three-dimensional velocity field from two-dimensional image-pair cross-correlation without reconstructing three-dimensional particle images.

1This work was supported by the Creative Research Initiatives (Diagnosis of Biofluid Flow Phenomena and Biomimic Research) of MEST/NRF of Korea.

5:32PM LC.00010 Accurate measurement of streamwise vortices in low speed aerodynamic flows, RYE M. WALDMAN, JUN KUDO, KENNETH S. BREUER, Brown University — Low Reynolds number experiments with flapping animals (such as bats and small birds) are of current interest in understanding biological flight mechanics, and due to their application to Micro Air Vehicles (MAVs) which operate in a similar parameter space. Previous PIV wake measurements have described the structures left by bats and birds, and provided insight to the time history of their aerodynamic force generation; however, these studies have faced difficulty drawing quantitative conclusions due to significant experimental challenges associated with the highly three-dimensional and unsteady nature of the flows, and the low wake velocities associated with lifting bodies that only weigh a few grams. This requires the high-speed resolution of small flow features in a large field of view using limited laser energy and finite camera resolution. Cross-stream measurements are further complicated by the high out-of-plane flow which requires thick laser sheets and short interframe times. To quantify and address these challenges we present data from a model study on the wake behind a fixed wing at conditions comparable to those found in biological flight. We present a detailed analysis of the PIV wake measurements, discuss the criteria necessary for accurate measurements, and present a new dual-plane PIV configuration to resolve these issues.

Monday, November 22, 2010 3:35PM - 5:19PM –
Session LD Turbulence Simulations II Long Beach Convention Center 102B

3:35PM LD.00001 Fully adaptive LES of homogeneous turbulent flows, GIULIANO DE STEFANO, Seconda Universita’ di Napoli, OLEG V. VASILYEV, University of Colorado at Boulder — With the recent development of wavelet-based techniques for computational fluid dynamics, adaptive numerical simulations of turbulent flows have become feasible. Adaptive wavelet methods are based on wavelet threshold filtering that makes it possible to separate coherent energetic eddies, which are numerically resolved, from residual background flow structures that are filtered out. The prescription of a given threshold for wavelet filtering directly links to the desired turbulence resolution. A new original strategy is presented for which the wavelet filtering threshold is not prescribed a-priori but determined on the fly for a given and known level of turbulence resolution. A completely adaptive eddy capturing approach that allows to perform variable fidelity numerical simulations of homogeneous turbulent flows is proposed. The new method is based on wavelet filtering with time-dependent thresholding that automatically adapts to the actual flow conditions in order to achieve the desired level of turbulence resolution. The filtered governing equations supplemented by a localized dynamic energy-based closure model are solved by means of the adaptive wavelet collocation numerical method.

3:48PM LD.00002 An hybrid a priori/a posteriori method for assessing LES models, DANIELE CARATI, BENJAMIN CASSART, BOGDAN TEACA, Universite Libre de Bruxelles — An hybrid approach combining the advantages of a priori and a posteriori methods is proposed for assessing the efficiency of LES models. The LES and the DNS are run simultaneously and an artificial forcing is used to maintain the LES field as close as possible to the filtered DNS field. Various diagnostics on this forcing are used to assess the quality of the LES model.

4:01PM LD.00003 A Subgrid Scale Estimation Model for Large Eddy Simulation1, KRISHNAN MAHESH, RAJES SAW, University of Minnesota — We propose a novel estimation procedure to model the subgrid velocity for Large Eddy Simulation (LES). The subgrid stress is obtained directly from the estimated subgrid velocity. The subgrid velocity is modeled as a function of resolved velocity (\(\overline{u_i}\)) and resolved strain-rate tensor (\(\overline{\Sigma}_{ij}\)). Using tensor invariants, we obtain an expression for subgrid velocity that is linear in \(\overline{u_i}\) and quadratic in \(\overline{\Sigma}_{ij}\) with three undetermined coefficients. These three coefficients are obtained by imposing the following constraints: (i) Galilean invariance, (ii) ensemble- averaged subgrid dissipation and (iii) subgrid scale kinetic energy. The subgrid dissipation may be obtained through either eddy–viscosity models or a new dynamic model for dissipation. The subgrid kinetic energy may be obtained either from the dynamic Yoshizawa model or a modeled transport equation. The estimation model is applied to isotropic turbulence and good results are obtained. Realistic backscatter is also predicted. We also extend the estimation procedure to LES of passive scalar transport and propose an estimation model for subgrid scale scalar concentration. The model is applied to decaying isotropic turbulence with an uniform mean scalar gradient and good results are obtained.

1Supported by the Office of Naval Research (ONR) under grant N00014–08–1–0433
4:14 PM LD.00004 Quantification of Sub-Grid-Scale Terms in Plasma Turbulence¹, Ting Rao, Columbia University, Ravi Samtaney, David Keyes, King Abdullah University of Science and Technology — The notions of DNS and LES are well established in hydrodynamic turbulence. We apply similar ideas to plasma turbulence whose governing equation (i.e., the Vlasov or Fokker-Planck equation) is higher dimensional (generally 6D). Analogously to the Navier-Stokes LES equations, we filter and derive sub-grid-scale terms for the Vlasov equation. It is known in gyrokinetic plasma turbulence that nonlinear interactions in velocity space lead small scales generation of the distribution function up to the collisional scales (see Schekochihin et al., Plasma Phys. Control. Fusion 2008). Because the gyrokinetic description still requires a 5D phase space description, and hence is computationally expensive, our first approach is to quantify the SGS terms in the context of drift kinetic equations in 4D phase space under the electrostatic approximation using high resolutions up to a billion grid cells. We examine the phenomenology of the SGS terms by a careful quantification using high-order DNS of drift kinetic turbulence. The eventual goal is to develop SGS models for use in under-resolved or LES of plasma turbulence.

1TR is supported by the Center for Plasma Edge Simulations, under a US-DOE grant. RS and DK are supported by KAUST. Simulations were undertaken on the Shaheen IBM Blue-Gen P at the KAUST Supercomputing Laboratory at KAUST.

4:27 PM LD.00005 Dynamic Lagrangian model and wall model for LES on unstructured grids¹, Amam Verma, Krishnan Mahesh, University of Minnesota — We discuss a dynamic Lagrangian averaging approach applied in conjunction with the standard dynamic model for large-eddy simulation. Unlike the conventional Lagrangian dynamic model where the Lagrangian time scale contains an adjustable parameter \( \theta \), we propose a dynamic time scale based on a “surrogate-correlation” of the Germano-identity error. The absence of any multi-linear interpolation makes this approach particularly suitable for unstructured grids. We also discuss a dynamic wall model obtained by incorporating RANS constraints into a dynamic SGS model. Unlike conventional approaches, Reynolds stresses are used as constraints on the mean SGS stress so that the constraining Reynolds stress closely matches the computed stress only in the mean sense. We use the Germano-identity error as an indicator of LES quality so that the RANS constraints are activated only where the Germano-identity error exceeds a certain threshold. These proposed models are applied to LES of turbulent channel flow at various Reynolds numbers and grid resolutions to obtain significant improvement over the dynamic Smagorinsky model, especially at coarse resolutions.

1This work was supported by the United States Office of Naval Research under ONR Grant N00014-08-1-0433.

4:40 PM LD.00006 Assessment of SGS models in an \( Re_c = 2000 \) channel flow¹, Sanjeeb Bose, Parviz Moin, Center for Turbulence Research, Stanford University — Typical validation of subgrid scale models for large-eddy simulation are performed at a relatively low Reynolds numbers and at reasonably fine resolutions. We assess the performance of subgrid scale models for large-eddy simulation at a high Reynolds number. Explicitly filtered large-eddy simulations of a fixed pressure gradient driven \( Re_c = 2000 \) channel flow are performed using the dynamic Smagorinsky, dynamic Vreman (You & Moin, 2007; Vreman, 2004), and a dynamic eddy viscosity model with \( r_t = Ck_{sgs}|S|^{-1} \). The resolution of LES simulations is chosen to be quite coarse (\( \Delta x^+ \approx 155, \Delta y^+ \approx 78 \)) in order to highlight the deficiencies of the subgrid scale models. Mean velocity profiles, rms fluctuations, and one dimensional energy spectra are compared for both DNS and filtered DNS (Hoyas & Jimenez, 2006). The \( k_{sgs}|S|^{-1} \) model most accurately predicts the mean velocity profile, predicts the mass flux within 1.5%, and the centerline velocity within 3%. The effect of using a global coefficient for the eddy viscosity model versus a wall normal varying model coefficient will also be discussed.

1Supported by the DOE Computational Science Graduate Fellowship and DOE PSAAP program

4:53 PM LD.00007 A new dynamic eddy viscosity model for LES¹, Roel Verstappen, University of Groningen, Sanjeeb Bose, Center for Turbulence Research, Stanford University, Jungil Lee, Haecheon Choi, Center for Turbulence and Flow Control Research, Seoul National University, Parviz Moin, Center For Turbulence Research, Stanford University — A new dynamic eddy viscosity model based on the geometric mean of the eigenvalues of the resolved strain rate tensor, \( r_t \sim \Delta^2(\lambda_1 \lambda_2 \lambda_3)^{1/3} \), is proposed. The model is derived from the formal construction of the minimal eddy viscosity that is required to guarantee that all scales smaller than the filter width, \( \Delta \), are dissipated. This dynamic eddy viscosity model correctly predicts the decay rate for decaying isotropic turbulence and the predicted energy spectra are in good agreement with filtered DNS results. The mean velocity profile and the rms fluctuations are also in good agreement with filtered DNS results in an \( Re_c = 500 \) channel flow using this model: It is shown that the eddy viscosity also obeys a \( y^3 \) scaling near the wall when the model coefficient is computed using the dynamic procedure of Germano et al. (1991). The eddy viscosity properly vanishes for laminar flows and at solid boundaries, even without the aid of the dynamic procedure.

1Conducted during the 2010 CTR Summer Program

5:06 PM LD.00008 On the physics of turbulent flows in natural meander bends: Insights gained by LES¹, Fotis Sotiropoulos, Seokkoo Kang, University of Minnesota — Turbulent flow in a natural meander bend with riffle-pool sequences and arbitrarily complex large-scale roughness elements is simulated using high-resolution LES. The complex stream bathymetry is handled with a new version of the curvilinear immersed boundary method capable of carrying out LES in arbitrarily complex geometries with the dynamic Smagorinsky model and wall modeling. The computational grid is fine enough to resolve vortex shedding from cm-scale roughness elements in the riffles. The computed results are compared with experimental data and are shown to be in good overall agreement. The simulated flowfields are analyzed to provide new insights into the structure and driving mechanisms of the inner and outer bank secondary flow cells, the effects of large-scale roughness on turbulence anisotropy and anisotropy-driven secondary flows in the riffles, and the structure and impact of recirculation regions along the inner bank of the bend. The simulated flowfields also underscore and clarify previously hypothesized linkages between flow patterns and experimentally documented streambed morphodynamics.

1This work was supported by NSF grants EAR-0120914 (as part of the National Center for Earth-Surface Dynamics) and EAR-0738726, and the University of Minnesota Supercomputing Institute.

Monday, November 22, 2010 3:35PM — 5:45PM
Session LE Instability and Turbulence: Shear Layers Long Beach Convention Center 102C

3:35PM LE.00001 Linear stability analysis of planar reacting shear layers¹, Yee Chee See, University of Michigan, Matthiahs Ihme — Non-premixed flames are controlled by the mixing of fuel and oxidizer. These flames are susceptible to instabilities, which arise from hydrodynamic and heat-release instability mechanisms. To characterize these instability modes and their effects on the flame evolution and mixing, a spatial linear instability analysis is performed. In this formulation, a flamelet-formulation is utilized, in order to account for effects of detailed reaction chemistry and variations in thermo-viscous properties. The resulting set of governing equations is solved by employing a matrix method with spectral discretization. By applying this analysis to a methane/air flame in a planar shear layer, effects of molecular transport and reaction chemistry on the flame stability are investigated, and model results are compared detailed computational simulations.
3:48PM LE.00002 Effect of carbon content on hypersonic shear layer instability. LUCA MASSA, University Of Texas at Arlington — Thermochemistry interacts with fluid-dynamic processes when kinetic and convective time-scales overlap. Non-linear parabolized Navier-Stokes equations are applied to the analysis of the instability and transition of carbon containing shear layers in hypersonic conditions. Linear parallel analysis shows an increase in growth rate of magnitude up to 20%, for selected oxygen to carbon molar fractions. Localized maxima in linear growth rates are obtained for temperature close to the characteristic vibrational temperatures of carbon dioxide and for oxygen to carbon fraction of two, indicating potential destabilizing effect of carbon chemistry at high Eckert numbers. An increase in farfield temperature leads to an increase in growth rate, which is more marked at low carbon content. Non parallel effects are primarily related to streamwise relaxation for conditions away from equilibrium. The integrated kinetic energy from the parabolized analysis show a considerable change of kinetic energy growth with carbon content, identifying carbon dioxide as a destabilizing factor and causing increase of around 50%. Energy transfers from kinetic to sensible and latent enthalpic modes are analyzed within the parabolized evolution.

4:01PM LE.00003 Evolution of the mean dynamics of a shear-wake flow. MARC BAMBERGER, JOSEPH KLEWICKI, University of New Hampshire — A shear-wake flow forms in the post-separation region downstream of a splitter plate dividing two boundary layers that have different freestream velocities. Thus, the upstream (in-flow) condition for the shear-wake is a well-bounded flow, while the downstream (out-flow) state is a two-stream shear layer. Recent studies have revealed that the mean momentum equation of the turbulent boundary layer admits a four layer structure, and that the three of these layers the mean viscous force (gradient of the mean viscous stress) is of leading order. Conversely, the mean dynamics of the two stream shear layer are dominated by inertial terms (mean advection and Reynolds stress gradient). In this presentation we report on an experimental investigation of the evolution of the mean dynamics of a shear-wake flow. Single and x-array hotwire measurements are acquired in a relatively large scale shear layer facility at a velocity ratio of two. Special attention is given to better understanding the processes leading to the attenuation of the viscous force to lower order, and the evolution of a two-signed mean vorticity distribution to one that contains a single sign.

4:14PM LE.00004 Is the growth rate of a turbulent mixing layer universal? 1. SAIKISHAN SURYANARAYANAN, RODDAM NARASIMHA, JNCASR, Bangalore — The controversy on the universality of asymptotic spreading rates in turbulent shear flows remains unresolved even in the context of the widely studied mixing layer. This fundamental issue has deep implications on modeling. A strong case for the existence of a regime of universal spreading rate for a 2D temporal mixing layer simulated by a repeated array of point-vortices is presented. In the most extensive set of point-vortex simulations, involving up to 10000 vortices, ensemble averages over up to 108 realizations, it is established that the growth rate of momentum thickness varies by less than 1% from a universal value of 0.0167 times the velocity differential in the linear overlap between the initial condition and the (out-flow) state is a two-stream shear layer. Recent studies have revealed that the mean momentum equation of the turbulent boundary layer admits a four layer structure, and that three of these layers the mean viscous force (gradient of the mean viscous stress) is of leading order. Conversely, the mean dynamics of the two stream shear layer are dominated by inertial terms (mean advection and Reynolds stress gradient). In this presentation we report on an experimental investigation of the evolution of the mean dynamics of a shear-wake flow. Single and x-array hotwire measurements are acquired in a relatively large scale shear layer facility at a velocity ratio of two. Special attention is given to better understanding the processes leading to the attenuation of the viscous force to lower order, and the evolution of a two-signed mean vorticity distribution to one that contains a single sign.

4:27PM LE.00005 Similarity in 2-D spatially developing and long shear layers. C. CARTON DE WIART, G. WINCKELMANS, C. BAILLY, P. CHATELAIN, Université catholique de Louvain (UCL) - IMM, F. THIRIFAY, Cenaeo, A. RHOSKO, Caltech — 2D shear layers are studied using a high accuracy vortex-in-cell (VIC) method. The case investigated is U2/U1 = 0.38, as in the Brown and Rhosko experiment. The inflow corresponds to a regularized vortex sheet with thickness momentum thickness $\theta_0 = \pi/4$ and $Re_{\theta} = 54$. It then grows and smoothly undergoes transition, through TS waves and then K-H instabilities, to a “turbulent shear layer” developed at $x = 500$. Two computational domains are used: $L_1 = 2500$ and $L_2 = 3500$. Various outflow conditions are also used with $L_1$. We focus on self-similarity: profiles of $U/U_1, \omega v$, etc. as a function of $y = y(x-x_0)$ (with $x_0$ virtual origin), and slopes $d\theta/dx$, etc. The results of the $L_1$ simulations agree well with each other; the region $x \in [1800, 2500]$ being affected by the outflow and thus dismissed. They also agree well with the results of the $L_2$ simulation, thus confirming the $L_1$ simulations validity. The region $x \in [2800, 3500]$ is dismissed in the $L_2$ simulation. A remarkable result is that we do not obtain one long region of self-similarity but, instead, multiple such regions: the region $x \in [1800, 2500]$ is self-similar in $[2100, 2600]$ with $d\theta/dx = 0.0180$ and $d\omega v \times (\Delta U) = 0.0135$, then the region $x \in [1400, 2100]$ with 0.0146 and 0.0115, then the region $x \in [2100, 2600]$ with 0.0177 and 0.0140 (thus almost identical to the first region, potentially hinting at a recurring pattern).


4:53PM LE.00007 On the thickness of the turbulent/nonturbulent interface in shear layers. CARLOS DA SILVA, RODRIGO TAVEIRA, IST/IDMEC Technical University of Lisbon, Portugal — In free shear flows the flow field can be divided into two regions: the outer region where the flow is irrotational and the inner region where the flow is turbulent. The two regions are separated by a sharp interface: the turbulent/nonturbulent (T/N) interface. The thickness of this interface has been observed to be between the order of the Kolmogorov or the Taylor micro-scale in several experimental and numerical works. We show that the thickness of the T/N interface is equal to the radius of the large scale vortices (LVS) nearby this region. Direct numerical simulations (DNS) of planar jets at Reynolds numbers ranging from $Re_\lambda = 60 - 140$ using different initial conditions, and DNS of shear free irrotational/isotropic turbulence shows that the mean shear and the Reynolds number affect the T/N interface thickness insofar as they define the radial dimension of the LVS near the T/N interface, thus defining its thickness.

1Supported by project RN/DRDO/4124.

1Supported by NSF, CTS0352710.
5:06PM LE.00008 Effects of resolution on the fine scale features in the far field of a turbulent planar mixing layer. OLIVER BUXTON, SYLVAIN LARDEAU, SYLVAIN LAIZET, BHARATHRAM GANAPATHISUBRAMANI, Imperial College London — The three-dimensional structure and behaviour of the rate of rotation and strain rate tensors is examined in the far field of a turbulent planar two dimensional mixing layer. The mixing layer is simulated using the incompact3d DNS code at Reynolds number based on inlet conditions of 1000. The study looks at the effect of spatial resolution on the length scales of strain dominated (dissipation) and rotation dominated (enstrophy) regions of the flow by filtering the data and interpolating it onto successively coarser grids. The length scales of these regions are observed by means of probability density functions and the topological evolution is characterized by the intermediate eigenvalue of the rate of strain tensor. Additionally, the structure and length scales of strain producing and enstrophy producing regions of the flow, and the effects of spatial resolution upon them, are investigated. The effects of spatial resolution upon the interaction between strain and rotation are observed by looking at the alignment angle between the vorticity vector and the eigenvectors of the strain rate tensor.

5:19PM LE.00009 Dynamic mode decomposition of turbulent cavity flows for self-sustained oscillation. JIN LEE, HONG BEOM PARK, HYUNG JIN SUNG, KAIST — Self-sustained oscillations in cavity flows are due to the unsteady separation of boundary layer at the leading edge. The dynamic mode decomposition is employed to analyze the unsteadiness in extracted modes without the explicit knowledge of evolution operator of the data. Two different data of the cavity flow with and without self-sustained oscillations have been analyzed possessing thin and thick incoming boundary layers. The ratios of the cavity depth to the momentum thickness (D/θ) are 40 and 4.5, and the cavity aspect ratio is L/D = 2. The dynamic modes extracted from the thick boundary layer show that both of the boundary layer structures and the internal disturbance generated due to the presence of cavity coexist with coincidence in frequency spectrum but with different wavenumber spectrum, whereas the structures of the thin boundary layer show complete coherence among them causing self-sustained oscillations. This result suggests that the hydrodynamic resonance causing self-sustained oscillations occurs when the upcoming boundary layer structures and cavity perturbations coincide not only of frequencies, but also of wavenumbers. The structures of cavity perturbations change with the cavity size and the upcoming momentum thickness. The effects of cavity dimensions and incoming momentum thickness are discussed for oscillations to be self-sustained.

5:32PM LE.00010 Time resolved measurements of the pressure field generated by vortex-corner interactions in a cavity shear layer1, XIAOFENG LIU, JOSEPH KATZ, Johns Hopkins University — A 2D open cavity shear layer flow, especially its interaction with the trailing corner of the cavity, was investigated experimentally in a water tunnel at a Reynolds number of 3.8 × 10^6. Time-resolved PIV with an image sampling rate of 4500 fps and a field of view of 25 × 25mm was used to simultaneously measure the instantaneous velocity, material acceleration and pressure distribution. The pressure was obtained by spatially integrating the material acceleration (Liu and Katz, Exp Fluids 41:227-240). A large database of instantaneous realizations enables detailed visualization of the dynamic changes to shear layer vortices, such as deformation, breakup and trapping as they impinge and climb over the cavity trailing corner. These phenomena dominate the high pressure fluctuations near the corner, e.g. formation of a pressure minimum as the vortex is trapped on top of the corner. Ongoing statistical analysis examines the turbulence variables, focusing in particular on pressure-velocity and pressure-rate-of-strain correlations and their impact on the TKE balance.

Monday, November 22, 2010 3:35PM - 5:45PM – Session LF Chaos and Fractals Long Beach Convention Center 103A

3:35PM LF.00001 On the atmosphere of a moving body1, JOHAN ROENBY, Center for Fluid Dynamics and Department of Mathematics, Technical University of Denmark, HASSAN AREF2, Center for Fluid Dynamics and Department of Physics, Technical University of Denmark — We have explored whether a rigid body moving freely with no circulation around it in a two-dimensional ideal fluid can carry a fluid “atmosphere” with it in its motion. Somewhat surprisingly, the answer appears to be “yes”. When the body is elongated and the motion is dominated by rotation, we demonstrate numerically that, indeed, regions of fluid follow the body in its motion. Since there is a double-island structure for the case of pure rotation, as already found by Morton and Darwin many years ago, we see the existence of an atmosphere for the moving body as an example of the stability of Kolmogorov-Arnold-Moser tori. Our observations were reported in Physics of Fluids 22 (2010) 057103. The presentation will include animations not published with the paper and some indications of further work.

1Supported by the Danish National Research Foundation
2Permanent address: Department of Engineering Science and Mechanics, Virginia Tech

3:48PM LF.00002 The phase portrait of aperiodic non-autonomous dynamical systems1, ANA M. MANCHO, CAROLINA MENDOZA, ICMAT, CSIC — Geometry has been a very useful approach for studying dynamical systems. At the basis are Poincaré ideas of seeking structures on the phase space that divide it into regions corresponding to trajectories with different dynamical fates. We present a methodology to build global Lagrangian descriptors for arbitrary time dependent flows based on the intrinsic geometrical and physical properties of trajectories. Our new Lagrangian descriptors are applied to flows with general time dependence as those in geophysics. They succeed in detecting simultaneously, with great accuracy, invariant manifolds, hyperbolic and non-hyperbolic flow regions. We analyze convenience of different descriptors from several points of view: regularity conditions requested on the vector field, rate at which the Lagrangian information is achieved and computational performance. Comparisons with other traditional methods such as Finite Time Lyapunov Exponents (FTLE) will be also discussed.

1Supported by the following grants: OCEANTECH-PIF06-059 (CSIC), I-Math C3-0104 and MICINN-MTM2008-03754. Thanks to CESGA for computer support with FINIS TERRAE.

4:01PM LF.00003 On the cost-effectiveness of mixing optimization, OLEG GUBANOV, LUCA CORTELEZZI, McGill University — We consider the problem of estimating the cost-effectiveness of an optimal mixer (Gubanov & Cortelezzi, J. Fluid Mech., vol. 651, 2010), a mixer able to generate a mixture with a desired level of homogenization over a wide range of operating conditions while minimizing the homogenization time and cost. We generate a family of optimal mixers by extending the formulation of the sine flow (Liu et al., Chaos, Solitons and Fractals, vol. 4, 1994). We derive the Fourier sine flow, an egg-beater type of flow, which stirs a mixture by blinking velocity fields whose profile is defined as a Fourier sine series. We generate the four lower-level mixers by truncating the Fourier representation of the velocity profile to one, two, three and four modes, respectively. We formulate a constrained optimization problem for the velocity profiles. We use the mix-norm (Mathew et. al, Physica D, vol. 211, 2005) as a cost function. We couple profile and protocol optimizations and solve the problem every time the velocity fields are blinked. We compare the homogenization times achieved by the mixers. We show that, unexpectedly, the homogenization time does not decrease monotonically with increasing power input. Our results indicate that mixing optimization is most cost-effective at lower power inputs, it should be avoided in the low-middle range and becomes less attractive for higher power inputs.
W and 36°W is remarkably close to an ideal profile that exactly cancels the change in planetary vorticity. Analytical solutions for steady currents.

4:27PM LF.00005 Trapping of Swimming Particles in Chaotic Fluid Flow, NIDHI KHURANA, JERZY BLAWZDZIEWICZ, NICHOLAS T. OUELLETT, Yale University — We computationally study the dynamics of active particles suspended in a two-dimensional chaotic flow. The point-like, spherical particles have their own intrinsic velocity, and can therefore break transport barriers (KAM tori) in the flow. Even a small amount of swimming significantly affects the mixing. However, small but finite values of the swimming speed can lead to a decrease in mixing efficiency, as swimmers can get stuck in traps that form near elliptic islands in the flow field. We study the statistics of trapping times and its effect on transport dynamics.

4:40PM LF.00006 Rayleigh-Taylor unstable, premixed flames: the transition to turbulence, ELIZABETH HICKS, ROBERT ROSNER, University of Chicago — A premixed flame moving against a sufficiently strong gravitational field becomes deformed and creates vorticity. If gravity is strong enough, this vorticity is shed and deposited behind the flame front. We present two-dimensional direct numerical simulations of this vortex shedding process and its effect on the flame front for various values of the gravitational force. The flame and its shed vortices go through the following stages as gravity increases: no vorticity; a flat flame front; long vortices attached to a cusped flame front; instability of the attached vortices and vortex shedding (Hopf bifurcation); disruption of the flame front by the shed vortices, causing the flame to pulsate; loss of left/right symmetry (period doubling); dominance of Rayleigh-Taylor instability over burning (torus bifurcation); and, finally, complex interactions between the flame front and the vortices. We measure the subsequent wrinkling of the flame front by computing its fractal dimension and also measure mixing behind the flame front by computing the finite-time Lyapunov exponents.

4:53PM LF.00007 Characterizing changes in topological entropy via break up of almost-invariant sets, PIYUSH GROVER, Mitsubishi Electric Research Laboratories, MARK STREMLER, SHANE ROSS, PANKAJ KUMAR, Virginia Tech — In certain two-dimensional time-dependent flows, the braiding of periodic orbits provides a way to analyze chaos in the system through application of the Thurston-Nielsen classification theorem (TNCT). We build upon our earlier work that showed the first application of the TNCT to braiding of almost-invariant sets (AIS): AIS in a fluid flow are regions with high local residence time that can act as stirrers or ‘ghost roots’. In the present work, we discuss the break up of the AIS as a parameter value is changed, which results in a sequence of topologically distinct braids. We show that, for Stokes’ flow in a lid-driven cavity, these various braids give good lower bounds on the topological entropy over the respective parameter regimes. Hence we make the case that a topological analysis based on spatio-temporal braiding of almost-invariant sets can be used for analyzing chaos in fluid flows.

5:06PM LF.00008 On the relationship between stretching and homogenization in chaotic Stokes flows, MOHSEN GEHISARIEHA, MARK STREMLER, Virginia Tech — It is well known that chaotic particle trajectories can be generated in laminar flows by deterministic, time-periodic velocity fields. The exponential stretching of material lines in these flows can be quantified using the ‘topological entropy’. This measure of chaos is useful because in some circumstances it can be predicted mathematically using very limited information about the flow. We consider the relationship between this stretching and the mixing produced in these flows, which we evaluate by considering homogenization of a passive scalar. We study two different time-dependent, two-dimensional Stokes flow systems as examples: a double-lid-driven cavity flow and a 3-rod stirring system in a cylindrical domain. We will discuss the correspondence between topological entropy and decay in the variance of scalar concentration for varying parameters in these flows.

5:19PM LF.00009 Eigenmode analysis of scalar transport in distributive mixing, PATRICK ANDERSON, Materials Technology, M.K. SINGH, MICHEL SPEETJES, Materials Technology, Eindhoven, MATERIALS TECHNOLOGY, EINDHOVEN COLLABORATION — In this study we explore the spectral properties of the distribution matrices of the mapping method and its relation to the distributive mixing of passive scalars. The spectral decomposition of these matrices constitutes a discrete approximation to the eigenmodes of the continuous advection operator in periodic flows. The asymptotic state of a fully-chaotic mixing flow is dominated by the eigenmode corresponding with the eigenvalue closest to the unit circle. This eigenvalue determines the decay rate; its eigenvector determines the asymptotic mixing pattern. The closer this eigenvalue value is to the origin, the faster is the homogenization by the chaotic mixing. Its magnitude can be used as a quantitative mixing measure for comparison of different mixing protocols. Eigenvalues on the unit circle are qualitative indicators of inefficient mixing; the properties of its eigenvectors enable isolation of the non-mixing zones. Results are demonstrated of two different prototypical mixing flows: the time-periodic sine flow and the spatially-periodic partitioned-pipe mixer.

5:32PM LF.00010 Diffusion of adiabatic invariants and mixing in Stokes flows, ALIMU ABUDU, DMITRI VAINCHTEIN, Temple University — We discuss a quantitative long-term theory of mixing due to scatterings on resonances in 3-D near-integrable flows. As a model problem we use the flow in the annulus between two coaxial elliptic counter-rotating cylinders. We illustrate that the resonance phenomena cause the jumps of adiabatic invariants and mixing. Our results show that the resulting mixing can be described in terms of a single 1-D diffusion-type for the probability distribution function. Parameters of the diffusion equation are defined by the averaged statistics of a single passage through resonance.

Monday, November 22, 2010 3:35PM - 5:32PM – Session LG: Oceanography III Long Beach Convention Center 103B

3:35PM LF.00001 Cross-equatorial flow of Antarctic Bottom Water and the complete Coriolis force, ANDREW STEWART, PAUL DELLAR, University of Oxford — Conservation of potential vorticity strongly constrains large-scale flows in the oceans. It resists fluid crossing the equator, because a large relative vorticity is needed to balance the change in the sign of the planetary vorticity between hemispheres. However, the Antarctic Bottom Water (AABW) successfully crosses the equator deep in the Atlantic off the coast of Brazil. Our theoretical and numerical study of the AABW uses multilayer shallow water equations that include the complete Coriolis force due to the horizontal and vertical components of the Earth’s rotation vector. The widely neglected horizontal component is most prominent in the weakly-stratified abyssal ocean at the equator. The horizontal component combines with topography to create an extra term in the potential vorticity that offsets changes in the planetary vorticity with latitude. The observed topography between 33°W and 36°W is remarkably close to an ideal profile that exactly cancels the change in planetary vorticity. Analytical solutions for steady currents show 50% increases in transport due to the complete Coriolis force, as confirmed by numerical simulations of unsteady flows using an energy and enstrophy preserving scheme.
J. Fluid Mech using an exact theory (Howard and Yu, 2007, vol. 613, pp. 209-234). Issues of matching the solutions at the boundaries of corrugated and flat bottom will be discussed. Of particular interest is Bragg scattering, or Bragg resonance, which occurs when the spacing of corrugations is close to an integer multiple $m$ of a half water wavelength. The primary $(n = 1)$ scattering has been studied for well over two decades, studies of those higher order ($m > 1$) cases, however, are few. This will be discussed and the results presented.

3:48PM LG.00002 Generation of a jet by a sphere descending in stratified fluids. HIDESHI HANAZAKI, HIROYASU YOSHIKAWA, TOHRU OKAMURA, Kyoto University — Flow around a sphere descending vertically at constant speeds in uniformly stratified fluids is investigated by numerical simulations. As observed in recent salt-stratified tank experiments for strongly stratified fluids at high Schmidt number, vertical thin round jets and “bell-shaped” structures along the jets were observed in the lee of the sphere. The bell-shaped structure was found to be the consequence of steady internal waves in the lee of the obstacle, and it was actually generated where the downward velocity was the maximum. Temporal density variation on the rear stagnation point of the sphere and on the vertical symmetry axis of the flow in the lee of the sphere give some hints on the generation mechanisms of the jet, including the significant effects of molecular diffusion of salt.

4:01PM LG.00003 Boundary Layer Effects on Internal Wave Generation in a Stably Stratified Fluid. LAUREN EBERLY, JULIE VANDERHOFF, Brigham Young University — Through a series of laboratory experiments we attempt to quantify internal wave generation due to flow over the rough topography of a continental slope. Although significant progress has been made in flow over rough topography, few experimental studies have been done where the topography is oriented at an angle to both the isobaths and flow. Laboratory investigation is critical as linear theory is not completely accurate in describing generated internal waves. The disparity between linear theory and physical observation is greatest when the wave amplitudes reach a critical level or when boundary layer separation occurs. Previous experimental work on barotropic bottom topography suggests that linear theory over predicts the amplitude of generated lee waves as it does not account for effects due to boundary layer separation. This study employs a series of experiments to analyze an approximately two-dimensional, stably stratified fluid undergoing tidal flow over a topographically rough, sloped shelf. The laboratory set up utilizes a corrugated slope towed through the fluid as the forcing mechanism behind internal wave generation. The waves are visualized using the Synthetic Schlieren technique. Results show decreased internal wave amplitude from that predicted by linear theory.

4:14PM LG.00004 Direct numerical simulations of flow over ridges in presence of waves and current. LONG CHAU, KIRAN BHAGANAGAR, University of Texas, San Antonio — In this talk we demonstrate Direct numerical simulation (DNS) as a robust and a valid tool to study fundamental physics for coastal problems. We focus on turbulent pulsatile flow over 3-D ridged surfaces, which are relevant for oceanographic problems. We consider different morphological surfaces to explore the differences in turbulence production, dissipative and transport mechanisms. The influence of ridge shape and the pulse frequency on the scaling of the drag is explored.

4:27PM LG.00005 Turbidity Currents in Meandering Channels. MOHAMAD NASR-AZADANI, MICHAEL ZOELLNER, ECKART MEIBURG, UC Santa Barbara — We consider continuous, particle-laden gravity currents flowing along sinusoidal submarine channels bounded by levees, with special emphasis on the sediment transport. We investigate these flows via highly resolved three-dimensional direct Navier-Stokes simulations, based on an immersed boundary representation of the channel topography. Results are reported from a parametric study that focuses on shear stress profiles along the channel bed, secondary flow structures in channel cross-sections, lateral overflow over the levees, and sediment deposition, as functions of the channel geometry, the flow parameters, and the particle settling velocity.

4:40PM LG.00006 Double-Diffusive and Gravitational Instabilities in Particle-laden River Outflows. PETER BURNS, ECKART MEIBURG, UC Santa Barbara — When a sediment-laden river flows into the salty ocean, various instabilities may arise. In an initially static environment, these instabilities can be due to either double-diffusive or gravitational effects. As a function of the governing Peclet numbers and the particle settling velocity, we investigate via linear stability analysis under which conditions each instability mode dominates, and when the modes coexist. We find that the settling velocity has a non-monotonic effect on the temporal instability growth rates. While small settling velocities can serve to increase the growth rate of the instability, larger settling velocities are found to reduce the growth rate.

4:53PM LG.00007 Exact solutions for scattering by a patch of finite amplitude periodic bottom topography. JIE YU, GUANGFU ZHENG, Department of Civil, Construction and Environmental Engineering, North Carolina State University — Scattering of water waves by undulating bottom topography commonly occurs in coastal oceans, influencing many processes such as sediment transport and underwater acoustic propagation. In this study, we examine the scattering of water waves by a patch of periodic bottom corrugations with large amplitude, using an exact theory (Howard and Yu, J. Fluid Mech., vol. 593, 2007, pp. 209-234). Issues of matching the solutions at the boundaries of corrugated and flat bottom will be discussed. Of particular interest is Bragg scattering, or Bragg resonance, which occurs when the spacing of corrugations is close to an integer multiple $m$ of a half water wavelength. The primary $(n = 1)$ scattering has been studied for well over two decades, studies of those higher order $(m > 1)$ cases, however, are few. This will be discussed and the results presented.

1Supported by National Science Foundation (Grants CBET-0756271 and CBET-0845957) is greatly acknowledged.

5:06PM LG.00008 Investigation of internal wave amplitude estimates through phase-space ray-tracing. JULIE VANDERHOFF, Brigham Young University — An effective way to track the propagation and refraction of internal waves is by applying the ray, or WKBJ, approximation and performing numerical ray-tracing. Wave amplitudes are easily obtained in a phase-space formulation of the ray-tracing, which also avoids the caustics that are typically present in both the spatial and spectral formulations. However, the resulting phase-space solution must be projected onto the spatial and spectral domain to obtain solutions in those domains. Initial estimates are made to assess the accuracy of these projections using a simple background that varies in one spatial dimension and time. Implications for the short oceanic internal waves, and for cases of higher dimensions, are discussed.

5:19PM LG.00009 Numerical Simulations of an Asymptotically Reduced Model of Anisotropic Langmuir Turbulence. ZHEXUAN ZHANG, GREG CHINI, KEITH JULIEN, EDGAR KNOBLOCH — “Langmuir turbulence” is a wind and surface-wave driven flow that is thought to dominate vertical transport in the ocean surface boundary layer. The characteristic occurrence of quasi-coherent counter-rotating vortical structures elongated in the wind direction renders Langmuir turbulence strongly anisotropic. Recently, an asymptotically exact reduced model of this flow was derived using multiscale analysis (Chini, Julien & Knobloch, GAFD 2009). The reduced PDEs go beyond strictly 2D (downwind invariant) formulations of the governing Craik–Leibovich (CL) equations by consistently incorporating the dominant 3D physical processes while continuing to average or filter certain fast, fine downwind-scale flow features. Here, pseudospectral numerical simulations of the reduced PDEs are performed to explore the dynamics and bifurcation structure of the reduced model.

1Funding from NSF CAREER Award 0348981 is gratefully acknowledged.

Monday, November 22, 2010 3:35PM - 5:45PM – Session LH GFD: General I Long Beach Convention Center 103C
3:35PM LH.00001 Absolute instability of gravity waves1, MADIHA AHMED, JEAN-MARC CHOMAZ, Laboratoire d’Hydrodynamique (LadHyX), Ecole Polytechnique - CNRS — Although large-scale internal gravity waves of finite amplitude are known to be unstable, they are frequently observed in the lee of topography. We propose an explanation for this paradox by showing that the instability of these waves is convective and not absolute. Hence, in the frame of the mountain, a localized initial perturbation gives rise to a wave packet that grows but is entrained downstream, eventually leaving the flow undisturbed. The evolution of one such localized perturbation of a uniform finite-amplitude gravity wave is computed using direct numerical simulation of the Navier-Stokes equation under the Boussinesq approximation. The wave packet’s edge velocity is determined by analyzing the amplitude of the response on spatio-temporal rays. This generic technique allows discrimination between the absolute and convective nature of the instability and suggests that transition to turbulence might occur due to the nonlinear evolution of absolute instability.

1 Funded by AXA Research Fund doctoral grant

3:48PM LH.00002 Transient growth on horizontal shear with vertical stratification, CRISTOBAL ARRATIA, JEAN-MARC CHOMAZ, LadHyX, Ecole Polytechnique-CNRS, SABINE ORTIZ, LadHyX - ENSTA — We report an investigation of the three-dimensional stability of an horizontal shear flow, the hyperbolic tangent velocity profile, in an inviscid, stably stratified fluid. A previous work by Delonc et al. (2007) shows that the most unstable mode for this flow is two-dimensional. However, for strong stratification, the range of unstable vertical wavenumbers widens proportionally to the inverse of the Froude number. This means that the stronger the stratification, the smaller the vertical scales that can be destabilized. This is consistent with the self-similarity found by Billant and Chomaz (2001). We extend here that previous result by computing the optimal perturbations that maximize the energy growth up to a time horizon $T$ as a function of the streamwise and spanwise wavenumbers. We concentrate on short optimization times in the strong stratification limit, where the Billant and Chomaz self-similarity is verified to hold. The gravity wave components of the perturbations are obtained by means of a Craya-Herring decomposition which, in the absence of shear, corresponds to an exact separation between gravity waves and vertical modes for the linear dynamics. Intense excitation of gravity waves due to transient growth of perturbations is found in a broad region of the wavenumber plane, gravity waves being eventually emitted away from the shear layer.

4:01PM LH.00003 Internal bores: An improved model via a detailed analysis of the energy budget, ZACHARY BORDEN, TILMAN KOBLITZ, ECKMART MEIBURG, UC Santa Barbara — Internal bores, or hydraulic jumps, arise in many atmospheric and oceanographic phenomena. The classic single-layer hydraulic jump model accurately predicts a bore’s behavior when the density difference between the expanding and contracting layer is large (i.e. water and air), but fails in the Boussinesq limit. A two-layer model, where mass is conserved separately in each layer and momentum is conserved generally, does a much better job but requires for closure an assumption about the loss of energy across a bore. Through the use of 2D direct numerical simulations, we show that there is a transfer of energy from the contracting to the expanding layer due to viscous stresses at the interface. Based on the simulation results, we propose a two-layer model that provides an accurate bore velocity as function of all geometrical parameters, as well as the Reynolds and Schmidt numbers. We also extend our analysis to non-Boussinesq internal bores to bridge the gap between the single and two-layer models.

4:14PM LH.00004 Stability of isolated Barchan dunes, ANTOINE FOURRIÈRE, FRANÇOIS CHARRU, IMFT, Université de Toulouse, 31400 Toulouse — When sand grains are entrained by an air flow over a non-erodible ground, or with limited sediment supply from the bed, they form isolated dunes showing a remarkable crescent shape with horns pointing downstream. These dunes, known as Barchan dunes, are commonly observed in deserts, with height of a few meters and velocity of a few meters per year (Bagnold 1941). These dunes also exist under water, at a much smaller, centimetric size (Franklin & Charru 2010). Their striking stability properties are not well understood yet. Two phenomena are likely to be involved in this stability: (i) relaxation effects of the sand flux which increases from the dune foot up to the crest, related to grain inertia or deposition, and (ii) a small transverse sand flux due to slope effects and the divergence of the streamlines of the fluid flow. We reproduced aqueous Barchan dunes in a channel, and studied their geometrical and dynamic properties (in particular their shape, velocity, minimum size, and rate of erosion). Using coloured glass beads (see the figure), we were then able to measure the particle flux over the whole dune surface. We will discuss the stability of these dunes in the light of our measurements.

4:27PM LH.00005 A Lagrangian description of the energetics of stably stratified turbulence, SEUNGBUM JO, KEIKO NOMURA, JAMES ROTTMAN, University of California, San Diego — The general equations describing the energetics of stratified flows have been derived previously for a fixed volume in the Eulerian frame. Here we consider the energetics of a fluid particle in a homogenous flow and develop appropriate equations in the Lagrangian frame. Comparison with Eulerian analysis is discussed. We then illustrate our results using Lagrangian statistics from DNS of homogeneous stably stratified shear flows which include decaying, stationary, and growing turbulence conditions.

4:40PM LH.00006 Regimes of Turbulent Rotating Convective, ERIC KING, JONATHAN AURNOU — Heat transport by thermal turbulence has been of interest to the fluid dynamics community for decades. Furthermore, turbulent convective motions are responsible for many of the observed features of planets and stars, such as magnetic field generation and atmospheric jet formation. In these flows, the influence of background rotation through the Coriolis force is thought to be paramount. We present an examination of the importance of rotation in turbulent Rayleigh-Bénard convection through heat transfer measurements in a collaborative suite of laboratory experiments and numerical simulations. There exist two separate heat transfer regimes: rapidly rotating and non-rotating. We argue that the dynamical regime of a given convection system is determined by the competition between the thermal boundary layer and the Ekman boundary layer. This boundary layer control hypothesis permits the formulation of a predictive scaling of the transition between heat transfer regimes, and reconciles a broad array of previously disparate convection studies. The experimental results are also shown to apply to numerical models of planetary dynamos.

4:53PM LH.00007 The Role of Viscosity Contrast on the Plume Structure and Dynamics in High Rayleigh Number Convection, SREENIVAS KR, VIVEK N. PRAKASH, Engineering Mechanics Unit, JNCASR, Bangalore India, JAYWANT H. ARAKERI, Dept of Mechanical Engineering, IISc, Bangalore, India — We study the plume structure in high Rayleigh number convection in the limit of large Prandtl numbers. This regime is relevant in Mantle convection, where the plume dynamics is not well understood due to complex rheology and chemical composition. We use analogue laboratory experiments to mimic mantle convection. Our focus in this paper is to understand the role of viscosity ratio, $U$, between the plume fluid and the ambient fluid on the structure and dynamics of the plumes. The PLIF technique has been used to visualize the structures of plumes rising from a planar source of compositional buoyancy at different regimes of $U$ (1/300 to 2500). In the near-wall planform when $U$ is one, a well-known dendritic line plume structure is observed. As $U$ increases ($U > 1$; mantle hot spots), there is a morphological transition from line plumes to discrete spherical blobs, accompanied by an increase in the plume spacing and thickness. In vertical sections, as $U$ increases ($U > 1$), the plume head shape changes from a mushroom-like structure to a “spherical-blob.” When the $U$ is decreased below one ($U$=1; subduction regime), the formation of cellular patterns is favoured with sheet plumes. Both velocity and mixing efficiency are maximum when $U$ is one, and decreases for extreme values of $U$. We quantify the morphological changes, dynamics and mixing variations of the plumes from experiments at different regimes.
5:06PM LH.00008 Tidal instability and magnetic field generation, PATRICE LE GAL, DAVID CÉBRON, WIEZTE HERRMAN, MICHAEL LE BARS, STÉPHANE LE DIZÉS, IRPHE - CNRS/Univ. Aix-Marseille, ECOTEUMENTS TOURNAING ET GEOPHYSICS TEAM — We are interested in the interaction of the elliptical instability and magnetic fields in liquid metal flows both on laboratory and planetary scales. We first discuss an experimental set-up that realizes an elliptical flow of Galinstan under an imposed field. The presence of a magnetic field is here of double interest. Elliptically excited flows are monitored through the magnetic fields they induce and the instability may be controlled by Joule damping. This study provides some new insight in the nonlinear stages of the elliptical instability. In a planetary context, it is likely that elliptical instability under imposed field occurs in the tidally deformed moon Io of Jupiter. We show how tidally excited flows may significantly deform the imposed field of Jupiter through an induction process. Finally, we also study whether tidally driven flows can be capable of generating and sustaining magnetic fields through the dynamo effect. We present a first numerical study on the possibility of tidally driven dynamo action in triaxial spheroids.

5:19PM LH.00009 Tidal instability in exoplanetary systems, DAVID CÉBRON, IRPHE, France, RIM FARES, LAM, France, PIERRE MAUBERT, IRPHE, France, CLAIRE MOUTOU, LAM, France, MICHAEL LE BARS, PATRICE LE GAL, IRPHE, France — Due to their observational method, many of the discovered exo-planets are massive gas giants called “hot Jupiters” orbiting rapidly very close to their stars. Because of this proximity, these celestial bodies (stars and planets) are strongly deformed by gravitational tides. Therefore, a certain number of them must be the site of an hydrodynamic instability, called the tidal instability. Starting from measured astrophysical characteristics of these systems (masses, orbit radius, eccentricity and period, spin velocity), we show that this instability is, as expected, present in some of the stars when the ratio of the planet orbiting period to the star spinning period is not in a “forbidden range.” In this case, the instability should drive strong flows in the different fluid layers of both bodies. These flows must be taken into account to model the bodies interiors and subsequent properties (synchronization, dynamo, zonal winds...). Of particular interest is the possibility of modifying the alignment of the rotation axes of stars and planets by this tidal instability.

5:32PM LH.00010 Universal scaling law for the aspect ratio of a pancake vortex in a rotating stratified medium, ORIANE AUBERT, MICHAEL LE BARS, PATRICE LE GAL, IRPHE, France, PHILIP S. MARCUS, UC Berkeley — The Great Red Spot of Jupiter and the meddies in the Atlantic Ocean are the most famous and puzzling examples of long-lived pancake-like anticyclones that take place in a rotating and stably stratified medium. To reproduce and study these vortices in the laboratory, we inject a volume of isodensity dyed fluid in a rotating linearly stratified layer of salt water. Due to the Coriolis force, the injected fluid rapidly forms a pancake vortex whose long term evolution is quantified using PIV measurements and image processing. Three different phases take place: a fast geostrophic adjustment, an axisymmetrization by viscous coupling with the outside and finally a very slow decrease of the motion while preserving the self-similar shape of the vortex. This last regime can be described using a simplified system of equations based on a geostrophic equilibrium, where the energy source maintaining the long-lived vortex is the density anomaly with the outside: the vortex persists as long as the density anomaly remains, maintained by internal recirculations. The non-diffusive version of the equations gives an analytical solution for the self-similar shape of the vortex and the evolution law for the aspect ratio for small Rossby numbers. These theoretical predictions are verified experimentally and also agree with published measurements for the meddies and Jupiter’s Great Red Spot.
4:14PM L.J.00004 Implementation of Active Noise Control in a Closed-Circuit Wind Tunnel1. MATTHEW KUESTER, EDWARD WHITE Texas A&M University — Closed return wind tunnels, such as the Klebanoff–Sarc Wind Tunnel (KSWT) at Texas A&M University, can provide relatively low freestream turbulence levels but include noise sources that do not exist in flight. This background noise, such as fan and motor noise, can adversely affect boundary-layer transition experiments if the frequencies are in the range of unstable Tollmien–Schlichting waves. Passive acoustic treatments eliminate most noise propagating downstream from the fan to test section in the KSWT, but measurements showed upstream-traveling tonal noise propagating from the fan into the test section. The present paper describes the implementation of the technique of direct forcing, where a known, steady-state sound field is generated in the test section. The sound field is controlled by the momentum transfer from the flow coming through the spoilers. A significant reduction in sound pressure level is recorded and is found to be very sensitive to small changes in fractal grid parameters. Wake and drag force measurements indicated that the spoilers increase the drag whilst having minimal effect on the lift.

4:27PM L.J.00005 Aero-acoustic performance of Fractal Spoilers. J. NEDIC, B.GANAPATHISUBRAMANI, C. VASSILICOS, Imperial College London, J. BOREE, L. BRIZZI, A. SPOHN, Institute Pprime, CNRS-ENSMAGenerelle de Poitiers — One of the major environmental problems facing the aviation industry is that of aircraft noise. The work presented in this paper, done as part of the OPENAIR Project, looks at the problem of noise reduction from three small commercial aircrafts which are susceptible to atmospheric attenuation, and would be deemed less offensive to the human ear. A total of nine laboratory scaled spoilers were looked at, seven of which had a fractal design, one conventionally porous and one solid for reference. All of the spoilers were mounted on a flat plate and inclined at 30° to the horizontal. Far-field, microphone array and PIV measurements were taken in an anechoic chamber to determine the acoustic performance and to study the flow coming through the spoilers. A significant reduction in sound pressure level is recorded and is found to be very sensitive to small changes in fractal grid parameters. Wake and drag force measurements indicated that the spoilers increase the drag whilst having minimal effect on the lift.

4:40PM L.J.00006 Investigation and Control of Flow-Induced Helmholtz Resonance. RUOLONG MA, SCOTT MORRIS, BRIAN CASTELLO, University of Notre Dame — Grazing flow over the orifice of a Helmholtz resonator can result in a self excited resonance inside the cavity. A practical example is automotive sunroof buffeting. The resonance is generated by the vortical-acoustic coupling between the instability of the shear layer cross the orifice and the Helmholtz mode of the resonator. A simplified model was developed based on a control volume momentum analysis, which allows the determination of equations of state and dissipation terms. The present paper focuses on understanding the interaction between the forcing and the acoustic characteristics of the Helmholtz resonator to improve the model, as well as modification of the orifice geometry and exterior grazing flow to minimize the cavity pressure fluctuation.

4:53PM L.J.00007 Diffraction of an acoustic wave by a cavitating hydrofoil1, YURI ANTIPOV. Louisiana State University — Diffraction of a plane acoustic wave from a curvilinear foil placed in an inviscid fluid is considered. The flow is irrotational and steady-state. The upper and lower boundaries of the foil, h1(x) and h2(x) satisfy the condition h1(x) << 1. Because of the hydrofoil profile, it may be partly cavitating or fully cavitating. The problem is linearized, and the boundaries of the foil are replaced by their projections on the real axis. It is found that at the fore point of a foil whose upper and lower boundaries are wet or at the rear point of the supercavity, the complex velocity ω±(z) has a 1/2-singularity. At the fore point of a foil whose one side is wet and another one is cavitated, the function ω±(z) has a 1/4-singularity. The at points of a wet foil, the function ω±(z) is bounded. The problem is solved in a closed form in terms of singular integrals. The unknown boundary of the cavity is recovered. Since the boundary of the cavity is known, and the fluid mechanics problem is linearized, the diffraction problem reduces to a system of singular integral equations which are solved by the method of orthogonal polynomials. Generalizations to the case of the nonlinear Tulin single-spiral-vortex model and a system of hydrofoils are discussed1.

The authors gratefully acknowledge the support of NASA and AFOSR through AFOSR grant FA9550-09-1-0361. MSK also acknowledges the support of NASA/ASEE through a NASA Aeronautics Fellowship.

5:06PM L.J.00008 Modeling of the Flow Field from Turbofan Nozzles with Porous Fan Flow Deflectors1, ANDREW JOHNSON, JUNTAO XIONG, FENG LIU, DIMITRI PAPAMOSCHOU, U.C. Irvine — Wedge-shaped fan flow deflectors have shown promise in reducing noise from turbofan-type nozzles. Porous deflectors have a particular advantage as they allow some flow through the wedge, thus preventing strong velocity gradients that can cause excess noise near the wedge base. Computational modeling of the resulting flow field is challenging because it is not feasible to grid the perforations of the deflector. Instead, we use a body-force term in the momentum equation that is applied locally in the vicinity of the deflector porous surface. The body-force term is calibrated based on experimental velocity measurements inside and outside the flaps forming the wedge, as well as computations on simplified two-dimensional models of the wedge. For a given wedge angle, the flow field is governed by the porosity of the wedge surface as well as the “illumination angle” of the perforation holes. The latter parameter is crucial for obtaining a uniform velocity distribution at the wedge base, and can be adjusted by varying the surface thickness or the aspect ratio of the perforation holes.

5:19PM L.J.00009 Application of Conformal Transformations to Velocity Sources in Cavity Aeroacoustics, NATHAN MURRAY, University of Mississippi - NCFA, LAURENCE UKELEY, University of Florida - REEF — The surface pressure fluctuations observed in open cavity flows are related to the velocity sources present in the shear layer spanning the cavity opening. The relationship between these velocity sources and the pressure fluctuations can be expressed by Poisson’s equation giving a functional description of ∇2p. This relationship can be cast in an incompressible form, 12∇2p = f(x, u), or in a compressible form, ∇2p = f(x, u, ρ). In either case, the source terms can be integrated to yield the resulting pressure at a point. In order to accomplish this integration for cavity flows, a conformal transformation is needed to project the cavity flow domain into a flat plane. Here, the method that relies on a hodograph transformation is examined for incompressible, smooth homogeneous cavity flow. The results provide a look at the regions of the shear layer which directly affect the pressure at various locations along the streamwise extent of the cavity. These regions are then compared to dominant POD modes of the shear layer velocity.

5:32PM L.J.00010 Modification of Instability Waves and Radiated Sound due to Heating a Compressible Mixing Layer1, DANIEL BODONY, University of Illinois at Urbana-Champaign — It is well known that heating a turbulent jet at constant velocity alters its sound field and, to a lesser extent, the turbulent statistics. For low speed jets (Uj/α < 0.7) heating increases the radiated sound while at higher speeds heating decreases the radiated sound. In both cases the turbulence root-mean-square levels increase by 10% relative to the unheated jet. The cause of the sound field change is not known. This work examines the early jet development by considering the modification of instability waves on a compressible mixing layer due to heating using calculations of the linearized Euler equations coupled to a multiple scale expansion analysis of the governing problem. The near-field instability wave solution is matched asymptotically to a globally valid acoustic field for a uniformly valid solution. It is found that for high-speed mixing layers the growth-and-decay cycle of the instability waves is altered by heating, leading to increased entropy fluctuations which are less efficient sound radiators relative to vortical fluctuations.

1Supported by the NASA Supersonics Program.
Specifically, using a multi-scale (MS-RBC) and low-dimensional model (LD-RBC) for modeling red blood cells (RBCs), we study the role of fibrinogen intercellular forces in the formation of rouleaux structures at low shear rates. In particular, both models verify that RBC aggregation into rouleaux determines non-Newtonian response and they also predict a non-zero yield stress whose value depends on the fibrinogen concentration.

1Supported by NIH R01HL094270.

3:48PM LK.00002 Effects of Fibrinogen on RBC Aggregation and Rouleaux Formation1, DMITRY FEDOSOV, Forschungszentrum Juelich, WENXIAO PAN, Pacific Northwest National Lab, BRUCE CASWELL, Brown University, GERHARD GOMPPER, Forschungszentrum Juelich, GEORGE KARNIADAKIS, Brown University — We employ dissipative particle dynamics (DPD) to study human blood rheology. Specifically, using a closed torus-like ring of only 10 large, hard DPD-particles previously employed to represent a colloidal suspension. Except for channel sizes comparable to RBC diameters, suspensions of LD-RBCs also capture the essential hydrodynamics of blood flow in vessels as faithfully as do suspensions of MS-RBCs. In particular, the LD-RBC is in agreement with the experimental data for the apparent viscosity of blood and its cell-free layer over a wide range of hematocrits.

1Supported by NIH R01HL094270.

4:14PM LK.00004 The Contribution of Red Blood Cell Dynamics to Intrinsic Viscosity and Functional ATP Release, ALISON FORSYTH, Princeton University, MANOUK ABKARIAN, Université Montpellier II, JIANDI WAN, HOWARD STONE, Princeton University — In shear flow, red blood cells (RBCs) exhibit a variety of behaviors such as rouleaux formation, tumbling, swinging, and tank-treading. The physiological consequences of these dynamic behaviors are not understood. In vivo, ATP is known to signal vasodilation; however, to our knowledge, no one has deciphered the relevance of RBC microheterogeneity to the functional release of ATP. Previously, we correlated RBC deformation and ATP release in microfluidic constrictions (Wan et al., 2008). In this work, a cone-plate rheometer is used to shear a low hematocrit solution of RBCs at varying viscosity ratios (λ) between the inner cytoplasmic hemoglobin and the outer medium, to determine the intrinsic viscosity of the suspension. Further, using a luciferin-luciferase enzymatic reaction, we report the relative ATP release at varying shear rates. Results indicate that for λ = 1.6, 3.8 and 11.1, ATP release is constant up to 500 s⁻¹; which suggests that the tumbling-tanktreading transition does not alter ATP release in pure shear. For lower viscosity ratios, λ = 1.6 and 3.8, at 500 s⁻¹ a change in slope occurs in the intrinsic viscosity data and is marked by an increase in ATP release. Based on microfluidic observations, this simultaneous change in viscosity and ATP release occurs within the tank-treading regime.

4:27PM LK.00005 Occlusion of Small Vessels by Malaria-Infected Red Blood Cells1, HUAN LEI, Brown University, DMITRY FEDOSOV, Forschungszentrum Juelich, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — We use dissipative particle dynamics (DPD) method to study malaria-infected red blood cells (i-RBC). We have developed a multi-scale model to describe both static and dynamic properties of RBCs. With this model, we study the adhesive interaction between RBCs as well as the interaction between the Plasmodium falciparum (PF)-parasitized cells and a vessel wall coated with purified ICAM-1. In this talk, we will discuss the effect of the PF-parasitized malaria cell on the flow resistance of the blood flow at different parasitemia levels. The blood flow in malaria disease shows high flow resistance as compared with the healthy case due to both the stiffening of the i-RBCs (up to ten times) as well as the adhesion dynamics. For certain sizes of small vessels, the malaria-infected cells can even lead to occlusion of the blood flow, in agreement with recent experiments.

1Supported by NIH R01HL094270.

4:40PM LK.00006 Molecular-detailed simulation of red blood cells in Stokes flows1, ZHANGLI PENG, QIANG ZHU, UC San Diego — The red blood cell (RBC) membrane consists of a lipid bilayer and a cytoskeleton. By coupling a multiscale approach of RBC membranes with a boundary element method (BEM) for the exterior and interior fluids, we developed a numerical capacity to relate the fluid-structure interaction of RBCs in Stokes flows with detailed mechanical loads inside its molecular architecture. Our multiscale approach includes three models: in the whole cell level, a finite element method (FEM) is employed to model the lipid bilayer and the cytoskeleton as two distinct layers of continuum shells; the mechanical properties of the cytoskeleton are obtained from a molecular-based model; the spectrin, a major protein of the cytoskeleton, is simulated using a constitutive model. BEM is applied to predict the exterior and interior Stokes flows, and is coupled with the FEM of the membrane through a staggered coupling algorithm. Using this method, we simulated the tumbling and tank-treading behaviors of RBCs in shear flows, and investigated the RBC dynamics in capillary flows. The structural deformation of the cytoskeleton and the interaction force between the lipid bilayer and the cytoskeleton are predicted.

1This work was supported by the National Heart, Lung, and Blood Institute under Award No. R01HL092793.
Our method will be compared with other accelerated boundary integral techniques for Stokes flow. The efficacy of the method will be demonstrated by the methodology is then presented for calculating the single and the double layer integrals in the resulting formulation using the Green’s function mentioned above.

unknown surface tractions rather than the unknown surface velocity as is common for problems with non-matched viscosities in interfacial flows. An efficient first present an alternative formulation of the boundary integral equation that allows the use of GGEM. In this formulation, we get a second kind integral for the a fast O(NlogN) solution technique for the Stokes flow boundary integral equation in an arbitrary geometry. The acceleration is achieved via the use of the particles in a microchannel for studying the margination of particles.

flow with an immersed boundary method. The lateral migration properties of many cells in Poiseuille flow have been investigated. We also have combined the TX 77204 — A spring model is applied to simulate the skeleton structure of the red blood cell membrane and to study the red blood cell rheology in Poiseuille structure. This algorithm leads to highly accurate energy conservation and consequently superior stability. Our model reproduces the experimentally observed cell shapes.

1Financial support from the Volkswagen Foundation is gratefully acknowledged.

19:19PM 5K.00009 Numerical simulation of cell/cell and cell/particle interaction in microchannels1 , TSORNG-WHAY PAN, LINGLING SHI, ROLAND GLOWINSKI, Department of Mathematics, University of Houston, Houston, TX 77204 — A spring model is applied to simulate the skeleton structure of the red blood cell membrane and to study the red blood cell rheology in Poiseuille flow with an immersed boundary method. The lateral migration properties of many cells in Poiseuille flow have been investigated. We also have combined the above methodology with a distributed Lagrange multiplier/fictitious domain method to simulate the interaction of the red blood cells and neutrally buoyant particles in a microchannel for studying the margination of particles.

We acknowledge the support of the NSF grant DMS-0914788.

5:32PM 5K.00010 Accelerated Boundary Integral Method in Non-periodic Geometries and Applications to Flowing Capsules and Cells , AMIT KUMAR, MICHAEL GRAHAM, University of Wisconsin-Madison — We present a fast O(NlogN) solution technique for the Stokes flow boundary integral equation in an arbitrary geometry. The acceleration is achieved via the use of the General Geometry Ewald Like Method (GGEM) for computing the Green’s function and its associated stress tensor in the geometry of interest. In this talk, we first present an alternative formulation of the boundary integral equation that allows the use of GGEM. In this formulation, we get a second kind integral for the unknown surface tractions rather than the unknown surface velocity as is common for problems with non-matched viscosities in interfacial flows. An efficient methodology is then presented for calculating the single and the double layer integrals in the resulting formulation using the Green’s function mentioned above. Our method will be compared with other accelerated boundary integral techniques for Stokes flow. The efficacy of the method will be demonstrated by the solution of several large scale test problems involving the flow of capsules and red blood cells in a slit geometry.

Monday, November 22, 2010 3:35PM - 5:45PM
Session LL Mini-Symposium on Microhydrodynamics of Lipid Bi-layer Membranes Long Beach Convention Center 202A

3:35PM 5K.00001 Dynamic Coarse-grained Modeling of Lipid Bilayer Membranes , PAUL ATZBERGER, UCSB — A coarse-grained approach capturing features at the level of of individual lipid molecules is presented for the study of dynamic phenomena related to bilayer membranes. The model takes into account molecular interactions between lipids, hydrodynamic coupling, and thermal fluctuations. The model is parameterized to have bending elasticity, compression moduli, and shear viscosity comparable to experimentally studied bilayer membranes. To carry-out simulation studies using the model, new stochastic computational methods are introduced based on fluctuating hydrodynamics. Using this approach, specific simulation results are presented which characterize how molecular level interactions contribute to bilayer mechanics (stiffness, tension, compressibility), bilayer rheology (shear viscosity, normal stress differences), and the mobility of bilayer embedded particles (single and pair diffusivity tensors). Applications of the bilayer model and computational methods to problems in cell biology are also discussed.

4:01PM 5K.00002 Composition dynamics in lipid bilayer membranes over long length and time scales , FRANK BROWN, UCSB — We present a stochastic phase-field model for multicomponent lipid bilayers, which explicitly accounts for the quasi-two-dimensional hydrodynamic environment unique to a thin fluid membrane immersed in aqueous solution. Dynamics over a wide range of length scales (from nanometers to microns) for durations up to seconds and longer are readily accessed and provide a direct comparison to fluorescence microscopy measurements in ternary lipid/cholesterol mixtures. Simulations of phase separation kinetics agree with experiment and elucidate the role of hydrodynamics in the coarsening process.

4:27PM 5K.00003 Single vesicle dynamics in various flows: Experiment versus theory , VICTOR STEINBERG, Weizmann Institute of Science — Dynamics of a single vesicle in shear, elongation, and general flows is investigated experimentally. Phase diagram of three vesicle dynamical states is obtained experimentally in both shear and general flows. The new control parameter, the ratio of the vorticity to the strain rate \( \omega / s \), allows following an experimental path, which scans across the whole phase diagram with a single vesicle. Surprisingly, all three states and transitions between them are obtained on the same vesicle and at the same viscosity of inner and outer fluids. We reveal the physical nature of the key dynamical state, coined by us trembling, which shows up in intrinsic shape instability on each cycle resulted in periodical bursting of higher order harmonics depending on the value of the control parameter proportional to \( \omega / s \). The dynamics of trembling state is compared with dynamics of a vesicle a time-dependent elongation flow, where the wrinkling instability was discovered, and similar features are identified. Quantitative comparison with recently proposed models and numerical simulations for vesicle dynamics is reviewed.

1We acknowledge the support of the NSF grant DMS-0914788.
Grant and 2009-2010 Research Professorship Award.

In this work, microtubules and rectangular microchannels are compared with 2D and 3D fluid flow simulation for testing their effectiveness. In these microdevices, the interaction between the electric currents and magnetic fields results in Lorentz forces. Electric potential is supplied in the electrodes that patterned on the opposite walls of the channel. The Lorentz forces can be used to propel, stir, mix and/or manipulate fluid flow in the channel. Many works are focused on the dynamics of superparamagnetic microrods tethered to a solid wall and driven by a precessing magnetic field. We identify two distinct regimes: at low driving frequencies, the response of the rod is synchronous whatever the inclination of the field. Above a characteristic frequency, two qualitatively different behaviors are distinguished, depending on the inclination \( \theta \). For small field inclinations, the response of the filament remains synchronous at all frequencies. Conversely, when the field inclination exceeds a critical value \( \sim 55^\circ \), the response becomes asynchronous, and the size of the rod follows a complex trajectory exhibiting three-dimensional back-and-forth patterns. A minimal model, neglecting the flexibility of the rod and the hydrodynamic interaction with the wall, captures the main features of both regimes. We thus show that the complex trajectory patterns are chiefly due to the geometrical nonlinearities in the magnetic dipolar coupling. The critical angle is itself set by a purely geometrical criterion, arising from the superparamagnetic nature of the rod.

4:35PM LL.00004 Vesicles in a shear and Poiseuille flows. CHAOUQI MISBAH, LSP, CNRS and Universite J. Fourier Grenoble I, France — Vesicles, capsules and Red Blood Cells (RBCs) under flow are subject to considerable attention from both the theoretical and experimental points of view. Understanding their motions and dynamics is essential both at the fundamental level as a branch of biocomplex fluids, and at the technological level, such as the lab-on-chip technologies, targeted drug delivery, and blood flow diseases. First, we discuss the dynamics of individual biomimetic vesicles and biological entities (RBCs) under a simple shear flow, and overview the current state of the knowledge. Comparison with available experimental work will be provided. We then discuss the non-trivial rheology of dilute vesicle suspensions and results from experiments involving oscillatory shear with non-zero mean shear rate. Finally, we address a longstanding puzzle in the blood microcirculatory research: why do red blood cells adopt a non-symmetrical shape (called slipper shape) even in a symmetric flow? Our work shows that the symmetric shape is unstable in flow conditions encountered in microvasculature. Moreover, by adopting a slipper shape, the RBC acquires higher flow efficiency than the symmetric (parachute) shape. The extension of this study to a collection of cells will be outlined.

5:19PM LL.00005 Deformation and stability of lipid membranes in electric fields. PETIA VLHOVSKA, Brown University — The challenges and recent advances in the theoretical modeling of lipid membrane dynamics in electric fields will be overviewed. Vesicle shapes and the stability and poration of lipid bilayers will be discussed in relation to the complex electromechanics of membranes. First, the lipid membrane is an insulating shell impermeable to ions. Second, it is essentially a two-dimensional incompressible-fluid sheet; under stress lipid membranes store elastic energy in bending, while membranes made of cross-linked polymers are more likely to be stretched and sheared. Third, lipid membranes are extremely soft and they are easily bent by the thermal noise. I will show how the dynamical coupling of interface charging, membrane deformation, lipid density redistribution, and fluid motion gives rise to rich and sometimes surprising behavior of lipid membranes in electric fields.

Monday, November 22, 2010 3:35PM - 5:45PM
Session LM Microfluids: General VI: Magnetic and Electric Fields Long Beach Convention Center 202B

3:35PM LM.00001 Topology Optimization of Regenerators for Magnetic Refrigeration. FRIDOLIN OKKELS, GRIGORIOS PANAGAKOS, DTU Nanotech, Department of Micro- and Nanotechnology, Technical University of Denmark, Denmark — We show a new method of topological optimization for regenerators in magnetic refrigeration systems. We optimize the microstructure of regenerator to improve the cycle efficiency. For the high-level implementation of topology optimization we use a level set method. The model has to be steady state, and therefore the refrigeration cycle has been re-formulated, using harmonically varying fields. The amplitude model nicely reproduces the results from direct simulation of the thermodynamic cycle, and initial results from the topology optimization are presented.

3:48PM LM.00002 Numerical Simulation on the Liquid Bridge Formation by the Applied Electric Pulse. JIN SEOK HONG, IN SEOK KANG, POSTECH — In this work, liquid bridge (LB) formation by the applied electric field is analyzed numerically. Numerical simulation captures the temporal behavior of liquid surface during the LB formation between a top plate and a bottom nozzle. Numerical results show the three stages of LB formation: interface elevation, impact/fast spreading and slow spreading/stabilization. The effect of the applied voltage pulse is also studied in terms of minimal electrical energy for LB formation. Non-linear behavior such as bubble trapping at the impact of liquid to plate is also captured and explained qualitatively. Grounded and floating plate is considered. The wetting criterion for LB formation is suggested and explained in terms of capillary pressure. The linear decrease of the final contact radius with the top plate contact angle is shown from the numerical results. In addition, the effects of the liquid properties on the dynamics are briefly discussed.

4:01PM LM.00003 Three-dimensional rotational dynamics of superparamagnetic microrods. MARC FERMIGIER, NAIS COQ, SANDRINE NGO, OLIVIA DU ROURE, DENIS BARTOLO, ESPCI ParisTech — We investigate experimentally and theoretically the dynamics of paramagnetic microrods tethered to a solid wall and driven by a precessing magnetic field. We identify two distinct regimes: at low driving frequencies, the response of the rod is synchronous whatever the inclination of the field. Above a characteristic frequency, two qualitatively different behaviors are distinguished, depending on the inclination \( \theta \). For small field inclinations, the response of the filament remains synchronous at all frequencies. Conversely, when the field inclination exceeds a critical value \( \sim 55^\circ \), the response becomes asynchronous, and the size of the rod follows a complex trajectory exhibiting three-dimensional back-and-forth patterns. A minimal model, neglecting the flexibility of the rod and the hydrodynamic interaction with the wall, captures the main features of both regimes. We thus show that the complex trajectory patterns are chiefly due to the geometrical nonlinearities in the magnetic dipolar coupling. The critical angle is itself set by a purely geometrical criterion, arising from the superparamagnetic nature of the rod.

4:14PM LM.00004 Effectiveness of Magnetohydrodynamics in Microdevices for Fluid Flow. YOGENDRA PANTA, WEI LIN, Youngstown State University — Magnetohydrodynamics (MHD) offers an elegant means to control fluid flow in micro- and nano-devices without a need for any mechanical components with the device. In the presence of an external magnetic field in a microchannel filled with ionic sample solutions, the interaction between the electric currents and magnetic fields results in Lorentz forces. Electric potential is supplied in the electrodes that patterned on the opposite walls of the channel. The Lorentz forces can be used to propel, stir, mix and/or manipulate fluid flow in the channel. Many works are reported about the MHD micro channel devices for various applications over the last thirty years, but there is still a need for better understanding of flow behavior in these microdevices. Also, there are insufficient studies of flow phenomenon under MHD in microtubules compared to rectangular cross sectioned microchannels. In this work, microtubules and rectangular microchannels are compared with 2D and 3D fluid flow simulation for testing their effectiveness. In presence and absence of external magnetic fields, an extensive parametric study was performed in order to find out the cross dependencies within the various experimental parameters. Numerical simulations were found in a good agreement with published data and experimental results.

Supported by the Danish Agency for Science, Technology and Innovation.

1Supported by the grant R01-2009-0083830 from National Research Foundation (NRF) of Korea, and by the BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.

4:14PM LM.00004 Effectiveness of Magnetohydrodynamics in Microdevices for Fluid Flow. YOGENDRA PANTA, WEI LIN, Youngstown State University — Magnetohydrodynamics (MHD) offers an elegant means to control fluid flow in micro- and nano-devices without a need for any mechanical components with the device. In the presence of an external magnetic field in a microchannel filled with ionic sample solutions, the interaction between the electric currents and magnetic fields results in Lorentz forces. Electric potential is supplied in the electrodes that patterned on the opposite walls of the channel. The Lorentz forces can be used to propel, stir, mix and/or manipulate fluid flow in the channel. Many works are reported about the MHD micro channel devices for various applications over the last thirty years, but there is still a need for better understanding of flow behavior in these microdevices. Also, there are insufficient studies of flow phenomenon under MHD in microtubules compared to rectangular cross sectioned microchannels. In this work, microtubules and rectangular microchannels are compared with 2D and 3D fluid flow simulation for testing their effectiveness. In presence and absence of external magnetic fields, an extensive parametric study was performed in order to find out the cross dependencies within the various experimental parameters. Numerical simulations were found in a good agreement with published data and experimental results.

One of the authors (YMP) would like to acknowledge the YSU-School of Graduate Studies and Research for providing support through 2009-2010 URC Grant and 2009-2010 Research Professorship Award.
4:27PM LM.00005 Interaction of two magnetic particles in a rotating magnetic field. TAE GON KANG, Korea Aerospace University, MARTIEN HULSEN, JAAP DEN TOONDER, PATRICK ANDERSON, HAN MEIJER, Eindhoven University of Technology — A three-dimensional direct simulation method was employed to solve flows with paramagnetic particles suspended in a non-magnetic fluid. The numerical scheme enables us to take into account both hydrodynamic and magnetic interactions between particles in a fully coupled manner, regardless of the shape of particles. As for the magnetic forces working on particles, the results obtained from our scheme are compared with those from the dipole-dipole interaction model. We confirm the critical angle separating the nature of magnetic interaction with the angle obtained by the point-dipole approximation. Dynamics of interacting two particles in a rotating field is investigated, demonstrating the capability of the method to tackle general problems. Chain dynamics is highly influenced by the Mason number, the ratio of viscous force to magnetic force. Below a critical Mason number, the chain of two particles rotates as a rigid body following the field, but with a phase lag. Above the critical Mason number, however, the chain rotates in overall sense but with an oscillatory motion on top of the rotation. It is also found that the magnetic susceptibility of particles is a factor with an influence on the chain dynamics. At one representative value of the susceptibility, we compared our numerical results with experimentally observed data.

4:40PM LM.00006 Electric-field-induced pattern formation in colloidal suspensions. JAE SUNG PARK, DAVID SAINTILLAN, University of Illinois at Urbana-Champaign — We use numerical simulations to investigate the long-time dynamics and pattern formation in semi-dilute suspensions of colloidal spheres in a viscous electrolyte under a uniform electric field. Dielectrophoretic interactions between particles occur as a result of Maxwell stresses in the fluid, and the dynamics under these interactions are analyzed in the thin Debye layer limit. Simulations in large-scale suspensions in a thin gap are performed with periodic boundary conditions in the directions perpendicular to the electric field. Results show the rapid formation of finite chains in the field direction, followed by a slow coarsening process by which chains coalesce into hexagonal sheets and eventually rearrange to form mesoscale cellular structures, in agreement with recent experimental observations. The effects of suspension volume fraction, electrode spacing and field strength on this phase transition are described, and a simple explanation for the observed wave number selection is proposed based on the analysis of interactions between two identical finite-length chains.

4:53PM LM.00007 Stretching Tethered Polymer by Traveling Electric Fields. HSIENT-HUNG WEI, YENG-CHIN LI, TEN-CHIN WEN, National Cheng Kung University — In this work, we theoretically explore the use of traveling wave electric fields in stretching a charged polymer chain whose one end is pinned on a surface. The simple elastic dumbbell model is employed to elicit the essentials of the stretching. In a simple sinusoidal field, the chain merely stretches and contracts back and forth with a zero cycle-averaged extension, as expected. In a traveling electric field, on the contrary, the chain can be periodically pulled by subsequent strokes of the field without being fully contracted, and therefore can exhibit some extension during a cycle. And yet, the chain will not stretch at all if the field travels too fast. The detailed response would depend on the Deborah number, the ratio of the elastic force to the stretch force, and alpha, the ratio of the traveling field speed to the characteristic electrophoretic velocity of the chain. We not only show how the stretching is characterized by these parameters, but also provide the criteria for realizing the stretching in terms of electrode dimensions, the chain size, and the strength and frequency of an applied traveling field. A possible application to molecular sensing is also discussed.

5:06PM LM.00008 Electrothermophoretic Flow in Microfluidic Reservoirs. XIANCHUN XIJIAN, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921, JUNJIE ZHU, SIRIAM SIRDHARAN, Department of Mechanical Engineering, Clemson University, Clemson, SC 29634-0921, USA, GUOQING HU, LMH, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China — Electrophoretic flow is an efficient technique for manipulating liquids and samples in microfluidic devices. However, there exists inevitable Joule heating in electrokinetic flow due to the liquid’s resistance to the electrical current. As such, both temperature rises and gradients are caused in the liquid, which has long been known to affect the electrokinetic fluid and sample transport within the fluid conduit such as a micro capillary or a microchannel. So far, however, no work has been done on Joule heating effects in microfluidic reservoirs that are the origins of all fluid and sample motions. In this talk we present an experimental and numerical study of electrokinetic fluid flow in microfluidic reservoirs. We demonstrate that fluid circulations can be induced by electrothermal effects inside the reservoir, which is potentially useful for trapping biomolecules or enhancing sample mixing.

5:19PM LM.00009 Electrosprinning of a viscous-capillary jet within dielectric liquid bath. GUILLAUME RIBOUX, Sevilla University — An experimentally characterization of the whipping motion of an electrified micro-jet of glycerine immersed within a liquid bath is carried out. In particular, the determination of the evolution of the frequency, the wavelength and the amplitude of the whipping oscillations as a function of the dimensionless parameters: the capillary number, the electrical Bond number and a residence to electrical relaxation time ratio. The presence of whipping requires threshold values of the three parameters to be reached. The electrified cone radius strongly depend on the capillary and electrical Bond numbers. The whipping behaviour, which depends on the capillary number but only weakly on the electrical Bond number, presents three different regimes: periodic, quasi-periodic or chaotic. Results showed that the wavelength and the frequency of the jet whipping depend strongly of the electrical Bond number. The phase velocity of the whipping is constant and proportional to the visco-capillary velocity. The detected whipping envelope showed self-similar behavior after appropriate normalization and evolved downstream as a 3/2 power law of the normalized distance.

5:32PM LM.00010 An efficient discretization of the Poisson-Boltzmann equation with applications to electrostatic force calculation. MOHAMMAD MIRZADEH, TODD SQUIRES, FREDERIC GIBOU, University of California Santa Barbara — We present a finite difference discretization of the non-linear Poisson-Boltzmann (PB) equation over complex geometries that has second order accuracy. The level-set method is adopted to represent the interface and Octree (in three dimensions) or Quadtree (in two dimensions) data structures are used to generate adaptive grids. Such an approach garantees that the finest grid resolution is located near the interface where EDL forms and creates very large electric field. Several numerical experiments are carried which indicate the second order accuracy both in the case of Dirichlet and Neumann boundary conditions in $L^2$ and $L^\infty$ norms. Finally, we use our method to study the electrostatic interaction of double layers between charged particles in an unbounded bulk electrolyte as well as in a channel where the channel width is of the order of Debye length.
3:48PM LN.00002 The Effect of Relative Submergence and Shape on the Wake of a Low-Aspect-Ratio Wall-Mounted body, SEYED MOHAMMAD HAJIMIRZAIE, CRAIG WOJCJK, JAMES BUCHHOLZ, The University of Iowa — Wall-mounted bodies in boundary layer flows are ubiquitous in marine and engineering applications and significantly enhance momentum and scalar transport in their vicinity. In this experimental study we evaluate the role of relative submergence (the ratio of flow depth to obstacle height) and shape on the wakes around four different wall-mounted obstacles. We consider four obstacle geometries: semi-ellipsoids with the major and minor axes of the base ellipses aligned in the streamwise and transverse directions, and two cylinders with matching aspect ratios D/H (where D is the maximum transverse dimension and H is the obstacle height). The aspect ratios considered are 0.67 and 0.89. DPIV was used to interrogate the flow. Streamwise structures observed in the mean wake include tip, base, and horseshoe vortex pairs as well as additional structures apparently not previously observed. The presence of a base vortex for such low-aspect-ratio obstacles is unexpected, and its strength increases with decreasing relative submergence. We will discuss hypotheses on the mechanisms of generation of the base and tertiary structures and their interconnection with the rest of the vortex skeleton.

4:01PM LN.00003 Experimental Investigations of the Flow past Circular Cylinders with Stepwise Discontinuities, CHRIS MORTON, SERHII YARUSEVYCH, University of Waterloo — Circular cylinders with step discontinuities in diameter are often encountered in engineering applications, e.g., finned tube heat exchangers and civil structures. This investigation is focused on wake vortex shedding from cylinders with one or two step discontinuities in diameter along the span. For uniform flow past a single step cylinder, the flow development is dependent on the Reynolds number (Re) and the ratio of the large cylinder diameter (D) to the small cylinder diameter (d), known as the diameter ratio (D/d). For a dual step cylinder, in addition to these two parameters, the aspect ratio of the large cylinder (L/D) is expected to influence vortex dynamics. This study has been performed for Re = 1050, D/d = 2, and a range of aspect ratios from 0.2 to 17. Experimental measurements have been acquired in a water flume facility using laser Doppler velocimetry and flow visualization. The results show that vortex shedding occurs in spanwise vortex cells of constant frequency. Vortex connections form between the spanwise vortices in these cells downstream of the step discontinuities, and vortex dislocations occur at cell boundaries. For a dual step cylinder, the aspect ratio is found to have a profound effect on vortex shedding, resulting in several distinct flow regimes.

4:14PM LN.00004 Horseshoe Vortex Dynamics in Cylinder-Wall and Cylinder-Endplate Junctions, ADAM BLACKMORE, TAYFUN AYDIN, ALIS EKMEKCI, University of Toronto — Unsteady horseshoe vortex systems forming near circular cylinder-wall and circular cylinder-end plate junctures are studied experimentally using Particle Image Velocimetry (PIV). Wall mounted cylinder is exposed to a flow with boundary layer thickness of 0.24D at the juncture, and the endplate mounted cylinder is exposed to flow that undergoes separation at the leading edge of the endplate and reattaches before the junction. PIV measurements on the cylinder-wall junction show presence of two primary horseshoe vortices. The vortex closest to the body (the first vortex) is periodically forced by the second vortex which develops further upstream. The trajectories of the horseshoe vortices on the plane of symmetry show that the second vortex shows several oscillations in the approach flow direction before merging closer to the first vortex, which also oscillates at the same frequency. The horseshoe vortex system at the cylinder-endplate junction shows quasi-periodicity over a certain leading edge distance, but has a more complex unsteady behaviour due to the separation/reattachment of the flow prior to the juncture.

4:27PM LN.00005 Secondary Vortex Structures in Vortex Generator Induced Flow, CLARA VELTE, VALERY OKULOV, MARTIN HANSEN, DTU — Passive rectangular vane actuators can induce a longitudinal vortex that redistributes the momentum in the boundary layer to control the flow. Recent experiments [1] as well as previous studies [2] have shown that a secondary vortex of opposite sign is generated along with the primary one, supposedly from local separation of the boundary layer due to the primary vortex. 2D flow visualizations of a vortex in the vicinity of a boundary support this hypothesis [3]. These secondary vortices are studied for various configurations – single generator, counter- and co-rotating cascades. The objective is to study their removal through cancelation in cascades using Stereoscopic Particle Image Velocimetry and flow visualization.


4:40PM LN.00006 Leading Edge Vortex Detection Using On-Body Pressure Sensing, JEFF DUSEK, JASON DAHL, MICHAEL TRIANTAFTYLOU, Massachusetts Institute of Technology, CENTER FOR ENVIRONMENTAL SENSING AND MODELING COLLABORATION — Ongoing experiments within the Center for Environmental Sensing and Modeling (CSENS) have shown that the low pressure region characteristic of a vortex allows for their detection and tracking using pressure sensors alone. While early experiments were conducted with wall mounted pressure sensors and externally generated vortices, a new series of experiments has succeeded in detecting separated flow generated by the sensing body. A combined pressure sensing and particle image velocimetry (PIV) approach was used to detect the leading edge vortex shed from a hydrofoil accelerated at a fixed angle of attack. A NACA 0018 foil was instrumented with four pressure sensors at discrete locations along the foil in the chord-wise direction. When accelerated from rest, the traces from each of the four pressure sensors displayed a distinctive, transient drop, consistent with results observed in previous experiments. From the pressure sensor results, it was theorized that a leading edge vortex was being created, and subsequently shed and convected along the foil chord. Two-dimensional PIV techniques were used to image the flow near the foil surface, allowing the anticipated vortex formation and shedding to be verified.

4:53PM LN.00007 Experimental evidence of 3D flows around corners at low Reynolds number, JOSUE SZNITMAN, DAVID CLIFTON, DEXTER SCOBEE, HOWARD STONE, ALEXANDER SMITS, Princeton University — Recently, Rusconi et al. (J R Soc Interface, 2010) have observed the formation of suspended filamentous biofilms in the middle plane of curved microchannels under laminar flow conditions. Motivated by such findings, we investigate experimentally the structure of 3D bounded viscous flows in the proximity of corners, at low Reynolds numbers (Re < 0.01). Beyond the location of the corner, shear driven flows are geometrically confined within rectangular-like channels of varying aspect ratios. Characteristic flows are experimentally generated using dynamic similarity in a toank filled with a highly viscous silicon oil. Quantitative flow measurements are obtained using PIV; regions of interest are interrogated by scanning a series of 2D planes in the vicinity of the corner. Past the corner, flows are predominantly uniaxial along the ductal length. However, near the corner, there exists a weak yet well-defined vortical flow structure; this secondary motion is experimentally observed via the deflection of PIV velocity vectors in the spanwise flow direction. Such flow structure yields a net steady-state flow focusing effect in the middle of the duct plane; the structure and strength of such focusing effect are discussed in the present talk.

5:06PM LN.00008 Dynamic Wind Loads and Vortex Structures in the Wake of a Wind Turbine, HUI HU, ZIFENG YANG, PARTHA SARKAR, Iowa State University — We report an experimental study to characterize the dynamic wind loads and evolution of wake vortex flow structures downstream of a horizontal axis wind turbine (HAWT). The experiments were conducted in a wind tunnel with a wind turbine model placed in a boundary layer flow developed over rough and smooth surfaces in order to study the effects of roughness and the resulting velocity and turbulence fields on the wake characteristics and fatigue loads acting on the wind turbine. In addition to measuring dynamic wind loads (both aerodynamic forces and moments) acting on the wind turbine model using a six-component load cell, a high-resolution Particle Image Velocimetry (PIV) system was used to make phase-locked flow field measurements to quantify the time-evolution of the wake vortex and turbulence flow structures shedding from wind turbine blades. The detailed flow field measurements were correlated with the wind load measurements to elucidate the underlying physics associated with turbine generation and fatigue loads acting on wind turbines.
was compared to a solution with no yeast cells. The impedance-based biosensor detects the change in impedance caused by the presence of yeast cells between experimental tests followed by finite element computer simulations were performed to selectively detect the quantity of yeast cells in a sample solution then

State University, NICHOLAS MATUNE, BENJAMIN MABBOTT, YOGENDRA PANTA, Youngstown State University — A microelectromechanical system

the unsteady behavior of the fluidic oscillator. Based on the results a new dimensionless number has been derived in order to characterize

fields. In the present study the working principle of the fluidic oscillator has been identified using three-dimensional unsteady RANS simulations and stability

necessary. Therefore fluidic oscillators can be used in extreme environmental conditions, such as high temperatures, aggressive media or within electromagnetic

aggregation, nanowire growth, electrochemical deposition, and biological interactions would benefit greatly from real-time, in-situ imaging with the nanoscale

microchannel walls that contain the target cells in a suspension medium. Microfluidic devices were fabricated by using two methods: traditional micromachining and photolithography for experimental purposes. An impedance analyzer was experimentally used for the measurement of the electrical impedance signals. Computer models based in COMSOL Multiphysics consisted of a long microchannel with two electrodes placed on opposite sides of the channel. Experimental data, simulation results and published data were compared and similar trends were found.

3:48PM LP.00002 Detection of Yeast Cells; Microfluidic Impedance Sensor11, KELSEY HULEA, Youngstown State University, NICHOLAS MATUNE, BENJAMIN MABBOTT, YOGENDRA PANTA, Youngstown State University — A microelectromechanical system (MEMS) based biosensor was proposed for the rapid detection of pathogenic bacteria and contaminants that pose a threat to public health. In this study, experimental tests followed by finite element computer simulations were performed to selectively detect the quantity of yeast cells in a sample solution then was compared to a solution with no yeast cells. The impedance based biosensor detects the change in impedance caused by the presence of yeast cells between the electrodes integrated into microchannel walls that contain the target cells in a suspension medium. Microfluidic devices were fabricated by using two methods: traditional micromachining and photolithography for experimental purposes. An impedance analyzer was experimentally used for the measurement of the electrical impedance signals. Computer models based in COMSOL Multiphysics consisted of a long microchannel with two electrodes placed on opposite sides of the channel. Experimental data, simulation results and published data were compared and similar trends were found.

1Youngstown State University - School of graduate studies and research

4:01PM LP.00003 Nanoaquarium for Imaging Processes in Liquids with Electrons1, JOSEPH GROGAN, HAIM BAU, University of Pennsylvania — The understanding of many nanoscopic processes occurring in liquids such as colloidal crystal formation, aggregation, nanowire growth, electrochemical deposition, and biological interactions would benefit greatly from real-time, in-situ imaging with the nanoscale resolution of the transmission electron microscopes (TEMs) and scanning transmission electron microscopes (STEMs). However, these imaging tools cannot readily be used to observe processes occurring in liquid media without addressing two experimental hurdles: sample thickness and sample evaporation in the high vacuum microscope chamber. To address these challenges, we have developed a nano-Hele-Shaw cell, dubbed the nanoaquarium. The device consists of a hermetically-sealed, tens of nanometers tall, liquid-filled chamber sandwiched between two freestanding, 50 nm thick, silicon nitride membranes. Embedded electrodes are integrated into the device for sensing and actuation. To demonstrate the device’s capabilities, we imaged diffusion-limited aggregation of 5nm diameter, gold nanoparticles. The rate of aggregation and the fractal dimension of the aggregate are consistent with light scattering measurements, indicating that the electron beam does not greatly alter the observed phenomenon.

1This work was supported, in part, by the NSF-NIRT (CBET 0609062).

4:14PM LP.00004 Numerical analysis of a fluidic oscillator, STEFAN HOETTGES, TORSTEN SCHENKEL, HERBERT OERTEL, Institute of Fluid Mechanics, Karlsruhe Institute of Technology (KIT) — The technology of fluid logic or fluidic has its origins in 1959 when scientists were looking for alternatives to electronics to realize measuring or automatic control tasks. In recent years interest in fluidic components has been renewed. Possible applications of fluidic oscillators have been tested in flow control, to reduce or eliminate separation regions, to avoid resonance noise in the flow past cavities, to improve combustion processes or for efficient cooling of turbine blades or electronic components. The oscillatory motion of the jet is achieved only by suitable shaping of the nozzle geometry and fluid-dynamic interactions, hence no moving components or external sources of energy are necessary. Therefore fluidic oscillators can be used in extreme environmental conditions, such as high temperatures, aggressive media or within electromagnetic fields. In the present study the working principle of the fluidic oscillator has been identified using three-dimensional unsteady RANS simulations and stability analysis. The numerical models used have been validated successfully against experimental data. Furthermore the effects of changes in inlet velocity, geometry and working fluid on the oscillation frequency have been investigated. Based on the results a new dimensionless number has been derived in order to characterize the unsteady behavior of the fluidic oscillator.
in understanding the flagellar waveform in migratory human sperm. The emergent waveform also induces curved swimming in an otherwise symmetric system, forces initiates an effective buckling behaviour, leading to a symmetry-breaking bifurcation that causes profound and complicated changes in the waveform and on the spermatozoon flagellum. For a wide range of physiologically relevant parameters, the nonlinear model predicts that flagellar compression by the internal studies, these models have been restricted to being geometrically linear or weakly nonlinear. In this talk, we study the effect of geometrical nonlinearity, focusing on the spermatozoon flagellum.

4:40PM LP.00006 Microfluidic waves for flow control. HOSSEIN HAJ-HARIIRI, MARCEL UTZ, University of Virginia, MATTY BEGLEY, University of California at Santa Barbara — The propagation of coupled waves in fluidic channels with elastic covers is discussed in view of applications for flow control in microfluidic devices. A theory is developed for pressure waves in the fluid coupled to bending waves in the elastic cover. At low frequencies, the lateral bending of the cover dominates over longitudinal bending, leading to propagating, non-dispersive longitudinal pressure waves in the channel. The theory addresses effects due to both the finite viscosity and compressibility of the fluid. The coupled waves propagate without dispersion, as long as the wave length is larger than the channel width. It is shown that in channels of typical microfluidic dimensions, wave velocities in the range of a few 10 m/s result if the channels are covered by films of a compliant material such as PDMS. The application of this principle to design microfluidic band pass and band stop filters based on standing waves is discussed. Characteristic frequencies in the range of a few kHz are readily achieved with quality factors above 30.

4:53PM LP.00007 A Portable, Air-Jet-Actuator-Based Device for System Identification, WAYNE STAATS, JESSE BELDEN, ANIRBAN MAZUMDAR, IAN HUNTER, MIT — System identification (ID) of human and robotic limbs could help in diagnosis of ailments and aid in optimization of control parameters and future designs. We present a self-contained actuator, which uses the Coanda effect to rapidly switch the direction of a high speed air jet to create a binary stochastic force input to a limb for system ID. The design of the actuator is approached with the goal of creating a portable device, which could be deployed on robot or human limbs for in situ deployment. The viability of the device is demonstrated by performing stochastic system ID on an underdamped elastic beam system with fixed inertia and stiffness, and variable damping. The non-parametric impulse response yielded from the stochastic system ID is modeled as a second order system, and the resultant parameters are found to be in excellent agreement with those found using more traditional system ID techniques. The current design could be further miniaturized and developed as a portable, wireless, on-site multi-axis system identification system for less intrusive and more widespread use.

5:06PM LP.00008 A hybrid molecular dynamics study of the translocation of DNA through entropic traps. PETR HOTMAR, Florida State University — The interplay between thermal diffusion and electrophoretic migration of λ-phage DNA in entropic traps was studied using a hybrid molecular dynamics algorithm. The governing systems of field equations are discretized by finite differences on curvilinear overlapping grids with the solvent modeled as a continuum in unsteady creeping flow. Similar to Brownian dynamics, the polymer segments are coarse-grained into a bead-spring model that follows Langevin dynamics. The hydrodynamic interactions are captured on a semi-empirical level with localized force-transfer. We have established the non-monotonic dependence of electrophoretic mobility on chain length, which characterizes the transition from the free flowing to the trapping behavior. We further quantify the subtle effects of dielectrophoresis and induced-charge electroosmosis on the polymer dynamics.

5:19PM LP.00009 Microfluidic assembly of multiscale soft materials, LIAN LENG, AXEL GUENTHER, University of Toronto — The vast majority of materials found in nature are characterized by length scales that span several orders of magnitude. Material properties such as porosity, permeability and elasticity are therefore locally and directionally tuned to their (biological) function and adapted to local environmental conditions. We use a massively scaled microfluidic approach to synthetically define multiscale complex fluids and soft materials with precisely tunable, non-isentropic bulk properties. Two or more fluids are continuously introduced to the device that consists of fifteen vertically bonded and fluidically connected substrate layers, and guided to an exit section that either consists of 23 equidistantly spaced channels or a 23 x 15 channel array. The flow rates through individual channels are computer-controlled. Upon entering a reservoir in a flow-focusing configuration, a spatially organized fluid with characteristic length scales of 250 microns and 10 mm was defined, and retained via a chemical reaction. To illustrate different soft material morphologies in one, two or three dimensions, we demonstrate the formation of isolated fibers (1D); planar graded and barcoded materials (2D); graded bulk materials and perfusable matrices (3D).

5:32PM LP.00010 Free-surface digital microfluidic systems for optimized SERS analysis in gas chromatography, BRIAN PIOREK, CHRYSAFIS ANDREOU, SEUNG JOON LEE, MARTIN MOSKOVITS, CARL MEINHART, University of California, Santa Barbara — A gas/liquid digital microfluidic platform was developed to chemically analyze gaseous analyte streams eluting from separation columns common in gas chromatography. The digital microfluidic stream is comprised of compartmentalized gaseous eluent packets segmented by SERS-active nanoparticles which are suspended within an aqueous phase. The microfluidic system is designed to optimize gaseous analyte transport into the SERS-active phase for chemical detection and analysis. Microfluidic and gas-phase flow patterns are controlled to produce reliable nanoparticle aggregation, resulting in SERS hot spots responsible for the PPT-level gas analysis mechanism. Since the flowing eluent is packetized by aqueous partitions, its spatiotemporal location can be controlled for effective analysis by SERS of nanoparticle hot-spot clusters occurring within individual partitions.

Monday, November 22, 2010 3:35PM - 5:45PM — Session LQ Bioluminiscence VII: Micro-Swimming III Long Beach Convention Center 203B

3:35PM LQ.00001 Nonlinear instability in flagellar dynamics: a novel modulation mechanism in sperm migration. H. GADELHA, E. GAFFNEY, Mathematical Institute, University of Oxford, D. SMITH, School of Mathematics, University of Birmingham, J. KIRKMAN-BROWN, School of Clinical and Experimental Medicine, University of Birmingham — Throughout biology, cells and organisms use flagella and cilia to propel fluid and achieve motility. While the mechanics of flagellum-fluid interaction has been the subject of extensive mathematical studies, these models have been restricted to being geometrically linear or weakly nonlinear. In this talk, we study the effect of geometrical nonlinearity, focusing on the spermatozoon flagellum. For a wide range of physiologically relevant parameters, the nonlinear model predicts that flagellar compression by the internal forces initiates an effective buckling behaviour, leading to a symmetry-breaking bifurcation that causes profound and complicated changes in the waveform and swimming trajectory, as well as the breakdown of the linear theory. The emergent waveform also induces curved swimming in an otherwise symmetric system, with the swimming trajectory being sensitive to head shape - no signalling or asymmetric forces are required. We conclude that non-linear models are essential in understanding the flagellar waveform in migratory human sperm.
3:48PM LQ.00002 Nature’s Helical Propeller. SAVERIO SPAGNOLIE, ERIC LAUGA, University of California, San Diego — Many microorganisms propel themselves through fluids by passing either planar waves (typically eukaryotes) or helical waves (typically prokaryotes) along a filamentous flagellum. Both from a biological and an engineering perspective, it is of great interest to understand the role of the waveform shape in determining an organism’s locomotive kinematics, as well as its hydrodynamic efficiency. In this talk we consider the specific issue of locomotion optimization for bacterial swimming, and we investigate the agreement between experimentally measured biological data on the swimming of E. coli and Salmonella, and optimization results from accurate numerical computations of the viscous flow fields around rotating bacterial flagella.

4:01PM LQ.00003 Flagellar propulsion near walls. ARTHUR EVANS, ERIC LAUGA, UC San Diego — Confinement and wall effects are known to affect the kinematics and propulsive characteristics of swimming microorganisms. When a solid body is dragged through a viscous fluid at constant velocity, the presence of a wall increases fluid drag, and thus the net force required to maintain speed has to increase. In contrast, recent optical trapping experiments have revealed that the propulsive force generated by human spermatozoa is decreased by the presence of boundaries. Here we use simple models to analytically elucidate the propulsive effects of a solid boundary on passively actuated filaments and model eukaryotic flagella. We show that in some cases, the increase in fluid friction induced by the wall can lead to a change in the waveform expressed by the flagella which results in a decrease of their propulsive force near a no-slip wall.

4:14PM LQ.00004 Hydrodynamics of insect spermatozoa. ON SHUN PAK, ERIC LAUGA, University of California San Diego — Microorganism motility plays important roles in many biological processes including reproduction. Many microorganisms propel themselves by propagating traveling waves along their flagella. Depending on the species, propagation of planar waves (e.g. *Ceratium*) and helical waves (e.g. *Trichomonas*) were observed in eukaryotic flagellar motion, and hydrodynamic models for both were proposed in the past. However, the motility of insect spermatozoa remains largely unexplored. An interesting morphological feature of such cells, first observed in *Tenebrio molitor* and *Bacillus rossius*, is the double helical deformation pattern along the flagella, which is characterized by the presence of two superimposed helical flagellar waves (one with a large amplitude and low frequency, and the other with a small amplitude and high frequency). Here we present the first hydrodynamic investigation of the locomotion of insect spermatozoa. The swimming kinematics, trajectories and hydrodynamic efficiency of the swimmer are computed based on the prescribed double helical deformation pattern. We then compare our theoretical predictions with experimental measurements, and explore the dependence of the swimming performance on the geometrical and dynamical parameters.

4:27PM LQ.00005 The dependence of the geometry efficiency of multi-flagellated bacteria on the geometric arrangement of the flagella. NOBUHIKO WATARI, University of Michigan, Macromolecular Science and Engineering Center, RONALD LARSON, University of Michigan, Department of Chemical Engineering — Multi-flagellated bacteria, such as *Escherichia coli*, often have flagella attached at random locations to the cell body. To study the effect of the number of flagella and the geometric arrangement of them to the swimming efficiency, we develop a simulation method using a bead-string model to account for the hydrodynamic and the mechanical interactions between multiple flagella and the cell body. First, a modeled bacterium is constructed using beads, which represent the hydrodynamic drag centers of the geometric elements of the bacterium. This modeled bacterium swims by rotating the flagella with constant torques at the bases of them. We have found that for modeled bacteria with two flagella, the swimming speed varies by 30% depending on the position of the base of the flagellum along the cell body, which affects the tightness of the bundling. We have also found that overly rigid flagella can slow migration by inhibiting flagellar bundling, since bundling requires some adjustment in flagellar shape to compensate for helical phase mismatch produced by irregular flagellar positioning. In general, by changing the geometric arrangement and the number of flagella, our simulation enables us to determine the optimal designing of a flagellated micro-swimmer.

4:40PM LQ.00006 “Corkscrew” vs. “tank-treading” propulsion of spirochetes. ALEXANDER LESHAN-SKY, Department of Chemical Engineering, Technion-IIT, Haifa, Israel, ODED KENNETH, Physics Department, Technion-IIT, Haifa, Israel — We consider the potential mechanism of spirochete propulsion driven by twirling of the outer cell surface coupled to counter-rotation of the helical body. We construct a proper slender body theory and use particle-based numerical approach allowing for modeling of locomotion in heterogeneous viscous environment. Depending on the helical pitch angle, two distinct propulsion gaits are identified: corkscrew-like locomotion, similar to propulsion powered by rotating helical flagellum, and surface tank-treading mode relying on hydrodynamic self-interaction of curved helical coils. The latter mechanism is closely related to the considered earlier propulsion of Purcell’s toroidal swimmer (Kenneth and Leshansky, Phys. Fluids 20, 063104, 2008). Significant augmentation of corkscrew propulsion gait in heterogeneous viscous medium anticipated from the numerical model is in accord with experimental observations of enhanced spirochete propulsion in polymer gels.

4:53PM LQ.00007 Parametric Studies of Swimming Filaments at Low Reynolds Numbers. TRISTAN SPOOR, STEPHAN KOEHLER, ERIC WILLISSON, Worcester Polytechnic Institute — Swimming of microorganisms in viscous fluids is a complex problem involving many degrees of freedom. In order to gain insight on this problem we investigate a simple model for locomotion of a thin, finite-length undulating filament. By keeping the number of parameters minimal we are able to compare similarities for four waveforms; two different sinusoidal (Cartesian and curvature), sawtooth, and square strategies. Beyond the domain of straight motion a number of turning strategies are also considered. These strategies are evaluated and in the case of the straight swimmers, although the differences between the strategies in terms of greatest speed are substantial, the differences in terms of greatest efficiency are small.

5:06PM LQ.00008 Dynamics of flagellar bundling. PIETER JANSSSEN, MICHAEL GRAHAM, University of Wisconsin-Madison — Flagella are long thin appendages of microscopic organisms used for propulsion in low-Reynolds environments. For *E. coli* the flagella are driven by a molecular motor, which rotates the flagella in a counter-clockwise motion (CCM). When in a forward swimming motion, all flagella bundle up. If a motor reverses rotation direction, the flagella unbundle and the cell makes a tumbling motion. When all motors turn in the same CC direction again, the flagella bundle up, and forward swimming continues. To investigate the bundling, we consider two flexible helices next to each other, as well as several flagella attached to a spherical body. Each helix is modeled as several prolate spheroids connected at the tips by springs. For hydrodynamic interactions, we consider the flagella to be made up of point forces, while the finite size of the body is incorporated via Faxén’s laws. We show that synchronization occurs quickly relative to the bundling process. For flagella next to each other, the initial deflection is generated by rotlet interactions generated by the rotating helices. At longer times, simulations show the flagella only wrap once around each other, but only for flagella that are closer than about 4 helix radii. Finally, we show a run-and-tumble motion of the body with attached flagella.

5:19PM LQ.00009 An Integrative Model of Hyperactivated Sperm Motility. SARAH OLSON, Tulane University, SUSAN SUAREZ, Cornell University, LISA PAUCI, Tulane University — Calcium (Ca²⁺) dynamics in mammalian sperm are directly linked to motility. These dynamics depend on diffusion, nonlinear fluxes, Ca²⁺-channels specific to the sperm flagellum, and other signaling molecules. The goal of this work is to couple Ca²⁺-dynamics to a mechanical model of a motile sperm within a viscous, incompressible fluid. An immersed boundary formulation of regularized Stokeslets is used to investigate the hydrodynamics and emergent waveforms and velocities. We will present recent progress on elements of this integrative model.
Studies of bubble shape, the effects of the Reynolds number (KANG, Inst. of High performance Computing — Taylor bubble rising is numerically investigated using a front tracking/finite difference method, with systematic

Stokes flow, the deformation of the capsule can be described by adapting existing conformal mapping techniques. We present analytical and numerical results for

MICHAEL BOOTY, MICHAEL SIEGEL, New Jersey Institute of Technology — We consider an inviscid bubble surrounded by an elastic membrane. In planar

influence the elongation of the tail and the wake structures, where higher Eo

properties are in quantitative agreement with experimental results.

Also, the bubbles take a larger eccentricity at the same positions. The Reynolds number of the bubbles is 372 and vortex shedding is observed. All dynamical

bubbles acquire elliptic shapes that oscillate

Hele-Shaw cell is accounted for in the model by including a brake term proportional to the corresponding component of the bubble centroid velocity in the

technique. The shape of the bubble is determined by the motion of the water surrounding the bubble, and the surface tension. The effect of the walls of the

bubble are calculated by solving a two-dimensional model that comprises the mass and momentum conservation equations coupled with a front tracking

study theoretically the dynamics of air bubbles ascending in a water filled Hele-Shaw cell. The bubble position and shape, and the flow inside and outside

of the bubble are calculated by solving a two-dimensional model that comprises the mass and momentum conservation equations coupled with a front tracking

technique. The shape of the bubble is determined by the motion of the water surrounding the bubble, and the surface tension. The effect of the walls of the

Hele-Shaw cell is accounted for in the model by including a brake term proportional to the corresponding component of the bubble centroid velocity in the

momentum conservation equation. The proportionality constant is a free parameter. We find that the bubbles follow a zigzag trajectory as they ascend. The

bubbles acquire elliptic shapes that oscillate ±45° around their geometrical center, with the largest inclination angle at the turning points of the zigzag motion.

Also, the bubbles take a larger eccentricity at the same positions. The Reynolds number of the bubbles is 372 and vortex shedding is observed. All dynamical

properties are in quantitative agreement with experimental results.

3:48PM LR.00002 Bubble dynamics in a Hele-Shaw cell — SAUL PIEDRA, EDUARDO RAMOS, CIE-UNAM — We study theoretically the dynamics of air bubbles ascending in a water filled Hele-Shaw cell. The bubble position and shape, and the flow inside and outside the bubble are calculated by solving a two-dimensional model that comprises the mass and momentum conservation equations coupled with a front tracking technique. The shape of the bubble is determined by the motion of the water surrounding the bubble, and the surface tension. The effect of the walls of the Hele-Shaw cell is accounted for in the model by including a brake term proportional to the corresponding component of the bubble centroid velocity in the momentum conservation equation. The proportionality constant is a free parameter. We find that the bubbles follow a zigzag trajectory as they ascend. The bubbles acquire elliptic shapes that oscillate ±45° around their geometrical center, with the largest inclination angle at the turning points of the zigzag motion. Also, the bubbles take a larger eccentricity at the same positions. The Reynolds number of the bubbles is 372 and vortex shedding is observed. All dynamical properties are in quantitative agreement with experimental results.

4:01PM LR.00003 Evolution of an elastic capsule in two-dimensional Stokes flow — MICHAEL HIGLEY, MICHAEL BOOTY, MICHAEL SIEGEL, New Jersey Institute of Technology — We consider an inviscid bubble surrounded by an elastic membrane. In planar Stokes flow, the deformation of the capsule can be described by adapting existing conformal mapping techniques. We present analytical and numerical results for the evolution and steady states of the capsule. These include the effects of different far-field conditions in the flow, resulting in behaviors such as tank-treading and cusp formation.

4:14PM LR.00004 Numerical Study of Taylor Bubble Dynamics — JING LOU, SHAOPING QUAN, CHANGWEI KANG, Inst. of High performance Computing — Taylor bubble rising is numerically investigated using a front tracking/finite difference method, with systematic studies of bubble shape, the effects of the Reynolds number (Re), the Weber number (We), and the Froude number (Fr). The thin liquid film thickness (w) and the wake length (lw). The effects of density ratio (\( \gamma \)), viscosity ratio (\( \lambda \)), Eötvös number (Eo) and Archimedes number (Ar) are examined in detail. The results show that the density ratio and the viscosity ratio have minimal effect on the dynamics of the Taylor bubble. Eötvös number and Archimedes number influence the elongation of the tail and the wake structures, where higher Eo and Ar result in longer lw. A critical value of unity of locally defined Weber number (\( W_e \)) is found to represent the sudden extension of the bubble tail. The Archimedes number drastically affects the final shape of Taylor bubble, the terminal velocity, the thickness of thin liquid film as well as the wall shear stress. A correlation between thin film thickness (\( w/lw \)) and Archimedes number (Ar) is obtained as: \( w/lw = 0.32 \lambda^{0.1} \).

\(^1\) The authors thank Professor Tapas Kumar Mandal from Indian Institute of Technology, Guwahati and Professor P. K. Das from Indian Institute of Technology, Kharagpur for their sharing their data during the course of this numerical study.

4:27PM LR.00005 Single bubble bouncing on a free surface and effect of the rising velocity and surface tension — TOSHIYUKI OYAMA, University of Tokyo, SHINTARO TAKEUCHI, SHU TAKAGI, YOICHIRO MATSUMOTO — The paper presents a numerical study of bubble bouncing on a free surface with a front tracking method. Contact time is determined as: the duration, within which the length between the mass center of the bubble and the initial position of the free surface is less than the initial bubble radius. The contact time is one of important values in the view of bouncing mechanism predicted by mass-spring modeling. The contact time is relational to the -0.5 power of surface tension coefficient as well as the prediction by mass-spring modeling. This result supports assumption that stored energy due to shape change is dominant to phenomenon of bubble bouncing rather than drainage between films. In the presentation we will refer the relation between the distance of two films between the free surface and the bubble and resolution.

4:40PM LR.00006 Direct numerical simulation of confined drops and bubbles at low capillary numbers — SHAHRIAR AFKHAMI, NJIT, ALEX LESHANSKY, Technion - Israel Institute of Technology, YURIKO RENARDY, Virginia Tech — The technology of microfluidics and its recent advance in the use of droplets within microchannels lead to the need for the understanding of drop deformation and breakup as a function of flow strength, physical parameters and fluid properties. We perform a combined asymptotic and numerical study of a simplified model for a dispersion which is pumped through a T-junction. The numerical method is based on a height-function formulation for a volume-of-fluid representation of the two liquids. Both the Bretherton and the Richardson bubble are used for benchmarking our code, thus establishing the degree of accuracy and robustness. The viscosity ratio of the drop to matrix liquid is varied to investigate the range of behavior from the presence of a bubble to that of a highly viscous drop. We focus on small Reynolds numbers and the limit of small capillary numbers where prior asymptotic theory of Leshansky and Pismen [Phys. Fluids 21, 023303 (2009)] has yet to be ascertained with computational results. In this regime, very strong capillary forces combined with confinement contribute to the difficulty in the direct numerical simulations. In particular, more than one mode of drop breakup has been observed in experimental data [Phys. Fluids 21, 072001 (2009)] when a large drop goes through a T-junction.

4:53PM LR.00007 ABSTRACT WITHDRAWN
Simulations of Bubble Motion in an Oscillating Liquid, A.M. KRAYNIK, L.A. ROMERO, J.R. TORCZYNSKI, Sandia National Laboratories — Finite-element simulations are used to investigate the motion of a gas bubble in a liquid undergoing vertical vibration. The effect of bubble compressibility is studied by comparing “compressible” bubbles that obey the ideal gas law with “incompressible” bubbles that are taken to have constant volume. Compressible bubbles exhibit a net downward motion away from the free surface that does not exist for incompressible bubbles. Net (rectified) velocities are extracted from the simulations and compared with theoretical predictions. The dependence of the rectified velocity on ambient gas pressure, bubble diameter, and bubble depth is in agreement with the theory. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

Multiscale modeling of bubbles rising in non-Newtonian fluids, ARTURO FERNANDEZ, North Carolina A&T State University — A multiscale method combining front-tracking with Brownian dynamics simulations is used to examine the dynamics of bubbles rising in a non-Newtonian fluid. Firstly, the evaluation of the material properties for the viscoelastic fluid will be discussed. Then, the results from the computations for a single bubble will be presented. We will discuss how the multiscale methodology is able to capture main features of the system dynamics including the appearance of a negative wake behind the bubble and the discontinuity in the terminal velocity.

Effect of contact line dynamics on capillary wave scattering by an infinitesimal barrier, LIKUN ZHANG, DAVID THIESSEN, Washington State University — Novel fluid configurations are possible at zero Bond number when capillary instabilities are countered by minimal solid support structures such as thin wires. Large aspect ratio liquid cylinders can be stabilized by an array of solid rings or by a helix. Capillary wave propagation on such channels involves scattering by a periodic array of barriers. The transmission and reflection of capillary waves by a single infinitesimal transverse barrier is considered theoretically by a matched evanescent wave expansion. An effective-slip boundary condition is applied for which the contact-line velocity is proportional to the deviation of the contact angle from its equilibrium value. Energy dissipation at the barrier is found to be most effective when the phase velocity is close to the phenomenological slip coefficient. The scattering of capillary waves on a liquid cylinder by a transverse ring agrees in the short-wave limit with the theory of gravity- capillary wave scattering by a transverse surface-piercing barrier in the limit of zero Bond number and zero barrier depth.

Contact-line pinning of a perfectly wetting volatile liquid overfilling a square hole with a tiny triangular groove along its perimeter: experiments and theoretical static shapes, YANNIS TSOMPAS, SAM DEHAECK, ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles, TIPs-Fluid Physics — Contact lines will stay pinned at sharp edges until they exceed a certain equilibrium angle as the Gibbs’s criterion indicates. In this preliminary study, we are trying to determine whether a groove can prevent a perfectly wetting liquid from spreading out of a square hole on the substrate. To this purpose, a highly-volatile electronic liquid has been chosen while the substrate was made of a polycarbonate plate with a square hole closely surrounded by a tiny groove of triangular cross-section. The results have shown this to act as an effective barrier, and the apparent contact angle (with respect to the horizontal) can attain considerable values. Moreover, the experiments indicated that the liquid drop adopts a shape with the apparent contact angle at the corners being much smaller than at other locations along the edge. This is explained on the basis of a simple static model including surface tension and gravity. The role of evaporation is also assessed.

4 A simple feedback algorithm to predict the contact angle with contact angle hysteresis, JUN KWON PARK, KWAN HYOUNG KANG, Department of Mechanical Engineering, Pohang University of Science and Technology (POSTECH), San 31, Hyoja-dong, Pohang 790-784, South Korea — In numerical analysis of contact-line problems, contact angle model plays an important role in predicting the motion of contact line. We developed a simple feedback algorithm to numerically predict the contact angle with considering the contact angle hysteresis. This algorithm automatically finds equilibrium contact angle which is between advancing and receding contact angle and the pinning position of contact line. We applied the numerical method to analyze the impacting droplet on a dry surface incorporating contact angle hysteresis and dynamic contact angle model. The numerical results showed good agreement with experimental data for the overall dynamics of the droplet, and the pinning process of contact line was also predicted well.

4 Contact angle hysteresis at the nanometer scale, THIERRY ONDARCUHU, MATHIEU DELMAS, MARC MONTIHOUS, CEMES-CNRS, NANOSCIENCES TEAM — Whereas thermodynamics predicts that the contact angle of a liquid droplet at rest on a solid surface is given by the Young-Dupré equation, experiments on real solid surfaces show that this contact angle is not universal, but exhibits a hysteresis which depends on the droplet history. Despite many theoretical and experimental studies devoted to this problem, a quantitative correlation between contact angle hysteresis and surface topography shape or repetition is still lacking. Here, we study the pinning of a contact line on nanometric defects by measuring, by atomic force microscopy, the capillary force exerted by a liquid on a carbon tip. This original quasi-1D geometry allowed us to study nanometric defects. We observe localized jumps which results from the pinning of the contact line on individual defects. A detailed study of the force curves validates the theory of weak and strong defects, postulated by Joanny & de Gennes (J. Chem. Phys., 81 (1984) 552). In particular, we bring the first experimental evidence of weak defects which do not contribute to hysteresis and study the energy dissipated on strong defects down to values of the order of kT. We then show how individual defects interact to give rise to contact angle hysteresis as measured at macrosopic scale, thus providing a precise description of the origin of contact angle hysteresis at nanometer scale.
4:27PM LS.00005 Optical characteristics and nonlinear dynamics of a pinned-contact double droplet system , JOSEPH OLLES, AMIR HIRSA, Rensselaer Polytechnic Institute, MICHAEL VOGEL, Voorhees, NJ — Through the use of an oscillating double droplet system (DDS), an adaptive varifocal liquid lens is created. Pinning the gas/liquid contact lines of the DDS eliminates viscous losses from moving contact lines and aids in simplifying the geometric parameter space. The use of millimeter scale diameters allows capillary effects to form stable droplets with spherical interfaces. We present a range of dependent parameters for the DDS as a fluidic lens which is made to oscillate by a sinusoidal pressure in millisecond cycles and various amplitudes. The lens is characterized through the following optical parameters throughout a cycle: sphericity, radius of curvature, focal length, and field stop. A non-linearity in the DDS’s resonant frequency is identified at various volumes; slight changes in the amplitude of the driving pressure produces a substantial shift in the resonant frequency of the system.

4:40PM LS.00006 Theoretical investigation of the motion of two-dimensional droplets on inclined substrates , NIKOS SAVVA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — We examine theoretically the dynamics of the motion of two-dimensional droplets on inclined topographical substrates. We take into account the effects of gravity and possible substrate vibrations. Assuming the presence of slip on the substrate and that the contact angle there always remains equal to its static value, the long-wave limit of the Stokes’ regime leads to a single equation for the evolution of the droplet thickness. Through a singular perturbation procedure, the flow in the vicinity of the contact points is asymptotically matched to the flow in the bulk of the droplet, to yield a set of integrodifferential equations for the location of the two droplet fronts. Our matching procedure is favourably compared with numerical solutions to the full problem. In the absence of vibrations, we find a substrate-induced hysteresis effect connected with the existence of a critical inclination angle beyond which the droplet can no longer be supported at equilibrium by the substrate. When substrate vibrations are present, we deduce criteria for the peculiar vibration-induced climbing of droplets reported in recent experiments.

4:53PM LS.00007 Capillary-Based Liquid Micro/Nano Droplet Deposition1 , ISKANDER AKHATOV, ARTUR LUTFURAKHMANOV, GREGORY LOKEN, DOUGLAS SCHULZ, North Dakota State University — Liquid droplet deposition through a capillary onto a substrate is studied. The application of pressure into the capillary causes a liquid meniscus to form at the outlet. Touching the substrate with the liquid meniscus and subsequent capillary retraction gives liquid deposition on the substrate. Theoretical and experimental studies of the steady liquid bridge structure between the capillary and substrate identified the range of parameters when deposition of small droplets (no blot) can be performed. Experiments revealed that in this range of parameters the size of deposited liquid droplets is less than 10-15% of inner diameter of the capillary. A logical next step in the demonstration of this approach is to translate from the microscale to the nanoscale using a nanocapillary navigated by a scanning tunneling microscope.

5:06PM LS.00008 Rapid wetting dynamics , ANDREAS CARLSON, GABRIELE BELLANI, GUSTAV AMBERG, Linné Flow Center, Department of Mechanics, The Royal Institute of Technology, LINNÉ FLOW CENTER, DEPARTMENT OF MECHANICS, THE ROYAL INSTITUTE OF TECHNOLOGY COLLABORATION — Contact lines between solids and liquid or gas interfaces appear in very many instances of fluid flows. This could be coffee stains, water-oil mixtures in oil recovery, hydrophobic feet of insects or leaves in nature. In the present work we elucidate some of the wetting physics governing the very rapid wetting. Experimental and numerical results of spontaneously spreading droplets are presented, where focus is directed towards understanding the very rapid flow regime and highly dynamic initial wetting phase, where the contact line speed is limited by dissipative processes on a molecular scale occurring at the contact line. In particular we show the influence of the surface wettability and the liquid viscosity on the spreading dynamics, such as the contact line motion and dynamic contact angle in time.

5:19PM LS.00009 Investigation of Contact Angle Behavior and Tilting Stability of Drops on Rough Surfaces1 , JASON SCHMUCKER, EDWARD WHITE, JOSHUA OSTERHOUT, Texas A&M University — A method for measuring full-field, instantaneous drop interface profiles on rough surfaces has been implemented to study contact angles on metallic surfaces with micron-scale roughness. Water drops measured span a range of Bond numbers from $Bo = 0.5$ to 5 on roughness in the range of $R_A = 0.8$ to 4.9. Experiments were conducted to provide data on contact angle variations about a single drop’s contact line and to investigate how contact angle depends on surface roughness. The method has also been used to study the stability of drops to sliding on tilted surfaces. Modifications of the contact line and distributions of contact angle are observed as surface angle is increased to the point of incipient sliding. The sensitivity of the stability parameters to the initial configuration of the drop is detailed, particularly in reference to initial contact line shape. Results such as critical inclinations and contact angles are discussed and compared with previous studies in the literature, beginning with Bikerman [J.Coll.Sci. 5:4, 1950] and including ElSherbini and Jacobi [J.Coll.Sci. 273, 2004].

5:32PM LS.00010 ABSTRACT WITHDRAWN

Monday, November 22, 2010 3:35PM - 5:45PM —
Session LT Bioluminvasion VIII: Flapping and Flying III Long Beach Convention Center Grand Ballroom B

3:35PM LT.00001 Effect of low-amplitude vibrations on impulsively-started wings1 , JESSICA SHANG, HOLGER BABINSKY, University of Cambridge — The development and shedding of leading edge vortices (LEVs) over wings is crucial to lift generation in the flapping flight of birds and insects. Many studies have investigated the flow field empirically by means of wing models that approximate or reproduce the wing kinematics. Wing models are often made of stiff materials (e.g. aluminum, steel) or are intentionally flexible to examine aerelastic properties. However, even stiff wings will vibrate under forces induced by accelerations, which may modify the flow field and the LEV shedding frequency. This study investigates the effects of start-up vibrations of impulsively started flat plates of different materials ($Re = 60,000$) at a post-stall angle of attack. Wing vibration was recorded with high-speed imaging and the flow field was analyzed with particle image velocimetry. Results do not eliminate the possibility of lock-on between the wing’s natural frequency and the LEV shedding frequency.

1NSF GRFP, Gates Cambridge Scholarship, US AFRL
3:48PM LT.00002 Investigation of vortex saturation for a simultaneously rotating and pitching wing. PRIYANKA MAHAJAN, MATTHEW RINGUETTE, JAMES TRZAKOS, JOHN SISTI, State University of New York at Buffalo. We investigate vortex saturation for a delta plate wing that simultaneously rotates and pitches. A vortex is saturated when it attains maximum circulation, which may occur at a non-dimensional time called the formation number. The experiment is done in a water tank and flow visualization is used to obtain the three-dimensional flow structure. We examine plate aspect ratios ranging from 2 to 4 with varying wing geometries and a Reynolds number of approximately 5000, similar to that of a hummingbird. We use digital particle image velocimetry (DPIV) to obtain the flow velocity, vorticity and circulation. The effects of spanwise (root-to-tip) flow and the tip vortex on the overall vortex structure are examined.

1Supported by the Air Force Office of Scientific Research Grant # FA9550-10-1-0281.

4:01PM LT.00003 Lift, Drag and Flow-field Measurements around a Single-degree-of-freedom Toy Ornithopter. RAMIRO CHAVEZ ALARCON, B.J. BALAKUMAR, JAMES ALLEN. The aerodynamics of a flight-worthy toy ornithopter under laminar inflow conditions are studied using a combination of load cell, flow visualization, high speed camera and DPIV experiments. All the experiments were performed in the large wind tunnel facility at New Mexico State University, with the exception of a free flight test of the model. Measurements from a six-axis load cell were used to capture the variation of the lift and drag forces at various angles of attack, flapping frequencies and free-speed velocities. Smoke visualization is used to clearly demonstrate that the momentum flux in the downward direction during downstroke exceeds the upward momentum flux during upstroke due to the flexion of the wing and its angle of attack. This net surplus creates the lift in such ornithopter designs despite the stroke symmetry. DPIV measurements are then performed at suitable locations to identify flow structures around the wing at various spanwise locations. A control volume analysis is performed to compare the momentum deficit in the wake to the load cell measurements.

1This work was funded in part by ARO grant W911NF-06-1-0487 and Continuum Dynamics Inc. STTR grant PO06-930.

4:14PM LT.00004 Three-Dimensional Wing Kinematics and Aerodynamic Characteristics of a Beetle in Free Flight. TIEN VAN TRUONG, DOYOU NG BYUN, HIEU TRUNG TRAN, TUYEN QUANG LE, HOON CHEOL PARK, Konkuk University, MINJUN KIM, Drexel University. Detailed three-dimensional wing kinematics and aerodynamic characteristics are experimentally presented for the free flight of a beetle, Alomyrina dichotoma. It is well known that there is quite a bit of flexibility in the beetle’s wing muscle system. An 8-DOF load cell was used to measure the forces and moments on both elytron and hind wing. Digital particle image velocimetry (DPIV) was used to obtain the flow velocity, vorticity and circulation. The effects of spanwise flow and stabilization of the leading edge vortices (Maxworthy, JFM 2007) are important, such as spanwise flow and stabilization of the leading edge vortices in hovering, forward, and climbing flight conditions. In addition, flow fields near regions of the elytron and the hind wing are quantitatively analyzed in order to visualize the LEV and calculate the circulation and lift coefficient by means of a DPIV experiment.

4:27PM LT.00005 Wing Flexion and Aerodynamics Performance of Insect Flights. HAIBO DONG, Wright State University, ZONGXIAN LIANG, YAN REN, Wright State University, WRIGHT STATE UNIVERSITY TEAM. Wing flexion in flapping flight is a hallmark of insect flight. It is widely thought that wing flexibility and wing deformation would potentially provide new aerodynamic mechanisms of aerodynamic force productions over completely rigid wings. There is ongoing work on developing fluid dynamics of freely flying insects due to the presence of complex shaped moving boundaries in the flow domain. In this work, we examine the aerodynamic performance of all four wings and understand the wake structures of such wings.

4:40PM LT.00006 Two- and three-dimensional numerical simulations of clap-fling-sweep of hovering insects. MARIE FARGE, LMD-CNRS, Ecole Normale Superieure, Paris, France, DMITRY KOLOMENSKY, M2P2-CNRS & Aix-Marseille University, France, KEITH MOFFATT, DAMTP, University of Cambridge, U.K., KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, France. The Lighthill–Weis-Fogh clap-fling-sweep mechanism is a movement used by some insects to improve their flight performance. As first suggested by Lighthill (JFM, 1973), this mechanism allows large scale oscillations in the wings to be established immediately as they start to move. Initially, the wings are clapped. Then they fling open like a book, and a non-zero circulation is established around each of them. Thus one wing can be considered as the starting vortex for the other. Then they sweep apart, carrying these bound vortices and generating lift. Since the insect wings have relatively low aspect ratio and rotate, three-dimensional effects are important, such as spanwise flow and stabilization of the leading edge vortices (Maxworthy, JFM 2007). To explore these effects, we perform direct numerical simulations of flapping wings, using a pseudo-spectral method with volume penalization. Comparing two- and three-dimensional simulations for the same setup clarifies the role of the three-dimensionality of the wake. Our results show that the two-dimensional approximation describes very well the flow during fling, when the wings are near, but three-dimensional effects become crucial when the wings move far apart.

4:53PM LT.00007 A computational study of the clap-and-fling aerodynamic mechanism. MARCOS VANELLA, University of Maryland, GRIGORIOS PANAGAKOS, Denmark Technological University, ELIAS BALARAS, University of Maryland — Clap-and-fling is a particular wing kinematic pattern utilized by some insects and birds to produce enhanced aerodynamic forces. It consists of two very distinct phases: i) the leading edges of the two wings are brought together near the upper limit of the upstroke and subsequently the wings are rotated around their leading edges, “clapping” like a closing book; ii) at the onset of the downstroke, and while they are still close, the two wings rotate around their trailing edges “flinging” apart. Prior theoretical and experimental work suggested that clap-and-fling is responsible for production of unusually high lift coefficients. However, due to limitations of the theoretical models and experimental techniques, detailed quantitative results are yet to be reported. In the present work we provide a concrete description of the underlying physics by means of high-fidelity three-dimensional simulations based on the Navier–Stokes equations for incompressible flow. Our results verify the lift enhancement trends observed in the experiments and indentify the particular flow patterns correlated with such increases.

1Research supported by AFSOR.
5:06PM LT.00008 Gliding flight in snakes: non-equilibrium trajectory dynamics and kinematics1, JAKE SOCHA, Virginia Tech, KEVIN MIKLASZ, Stanford University, FARID JAFARI, PAVLOS VLACHOS, Virginia Tech — For animal gliders that live in trees, a glide trajectory begins in free fall and, given sufficient space, transitions to equilibrium gliding with no net forces on the body. However, the dynamics of non-equilibrium gliding are not well understood. Of any terrestrial animal glider, snakes may exhibit the most complicated glide patterns resulting from their highly active undulatory behavior. Our aim was to determine the characteristics of snake gliding during the transition to equilibrium. We launched “flying” snakes (Chrysopelea paradisi) from a 15 m tower and recorded the mid-to-end portion of trajectories with four videocameras to reconstruct the snake’s 3D body position. Additionally, we developed a simple analytical model of gliding assuming only steady-state forces of lift, drag and weight acting on the body and used it to explore effects of wing loading, lift-to-drag ratio, and initial velocity on trajectory dynamics. Despite the vertical space provided to transition to steady-state gliding, snakes did not exhibit equilibrium gliding and in fact displayed a net positive acceleration in the vertical axis.

1We thank DARPA (grant W911NF100040) and National Geographic Television for their support.

5:19PM LT.00009 Flying Snake Flight Performance: Role of Cross-sectional Shape and Orientation of Tandem Body Segments, DANIEL HOLDEN, PAVLOS VLACHOS, Mechanical Engineering, Virginia Tech, JAKE SOCHA, Engineering Science and Mechanics, Virginia Tech — The “flying” snake (Chrysopelea paradisi) possesses one of the most dynamically complex gliding flight patterns found in nature. Unlike other airborne animals that possess wings or flaps, this species lacks appendages to aid in controlling its flight path and producing lift. While gliding, the snake undergoes a high-amplitude undulatory motion, during which it expands its ribs to double its body width so that its cross section mimics the shape of a thin airfoil with camber and leading and trailing edges. The goal of this study was to determine the aerodynamic forces produced by the snake and investigate the underlying fluid dynamics that give the snake its unique gliding and maneuvering capabilities. Two-dimensional force measurements and CFD simulations were performed on an anatomical model of the snake’s cross section to determine the steady and unsteady lift and drag coefficients, as well as the vortex shedding characteristics. These results show that the lift and drag produced by the model is dependent on Reynolds number, angle of attack, and the orientation of upstream and downstream body segments. Several tandem model configurations produced significant increases in lift and decreases in drag.

5:32PM LT.00010 Numerical investigation of wake structures of slow-flying bats, SHIZHAO WANG, XING ZHANG, GUOWEI HE, Institute of Mechanics, Chinese Academy of Sciences — Recently, some unique features of wake structure in bat flight have been revealed by experiments. It is found that the flow structure of bat flight is more complex than that of bird. A conceptual wake model of bat flight has been “rebuilt” using 2D DPIV images, but there is some risk of missing the details regarding dynamics of 3D vortex structures. Detailed flow information is still needed to understand the unsteady flow in bat flying. In this work, we perform 3D simulation of bat flying at the Reynolds number of 1000 (based on upstream flow and mean chord length) using the immersed boundary method. The geometry and wing-beat kinematics of bat are taken from the work of Watts et al. (2001). The topology and evolution of the wake structures are described. The variation of topology in wake structures with the flapping Strouhal number is investigated. Moreover, the link between the generation of high lift and leading edge vortex is also studied.

Monday, November 22, 2010 3:35PM - 5:45PM – Session LU Reacting Flows II Hyatt Regency Long Beach Regency A

3:35PM LU.00001 Large Eddy Simulations of A Turbulent Auto-Igniting C2H4 Flame DNS, EDWARD KNUDSEN, SHASHANK, Stanford University, HEINZ PITSCH, RWTH Aachen, ED RICHARDSON, JACKIE CHEN, Sandia National Laboratories — Large eddy simulations of a turbulent auto-igniting flame are performed to analyze the interaction of different combustion regimes in a flamelet modeling framework. The case that is considered is a direct numerical simulation (DNS) of a non-premixed jet flame at Re=10,000 with heated co-flow. This DNS was performed by Yoo et al. (Proc. Comb. Inst., 2010) using 1.29 billion cells and a 22 species mechanism. A series of flamelet-type approaches are applied in successive large eddy simulations of the flame to understand the importance and interaction of dissipation and auto-ignition. Simulations are first performed by relaxing either on purely 0-D auto-ignition chemistry or on purely steady non-premixed flamelet chemistry. These simulations significantly under- and over-predict the lift-off height, respectively. Two approaches are then considered that simultaneously account for these processes: a well known tabulated unsteady flamelet formulation and a multi-regime formulation that combines the limiting steady and auto-ignition solutions according to a regime indicator. Comparisons with the DNS demonstrate that these approaches lead to improved lift-off height predictions.

3:48PM LU.00002 Spark Ignition: Effects of Fluid Dynamics and Electrode Geometry, SALLY BANE, JACK ZIEGLER, JOSEPH SHEPHERD, California Institute of Technology — The concept of minimum ignition energy (MIE) has traditionally formed the basis for studying ignition hazards of fuels, and standard test methods for determining the MIE use a capacitive spark discharge as the ignition source. Developing the numerical tools necessary to quantitatively predict ignition is a challenging research problem and remains primarily an experimental issue. In this work a two-dimensional model of spark discharge in air and spark ignition was developed using the non-reactive and reactive Navier-Stokes equations. The simulations were performed with three different electrode geometries to investigate the effect of the geometry on the fluid mechanics of the evolving spark kernel and on flame formation. The computational results were compared with high-speed schlieren visualization of spark and ignition kernels. It was found that the electrode geometry had a significant effect on the fluid motion following spark discharge and hence influences the ignition process and the required spark energy.

4:01PM LU.00003 Critical Ignition in rapidly expanding self-similar flows1, MATEI I. RADULESCU, BRIAN MCN. MAXWELL, University of Ottawa — The generic problem of ignition of a particle undergoing an expansion given by a power law rate of decay behind a decaying shock is addressed in the present study. It is demonstrated, using a one-step Arrhenius irreversible reaction, that a sufficiently strong expansion wave can quench the reaction. The critical conditions for extinction are obtained in closed form in terms of the time scale for the expansion process and the thermo-chemical properties of the gas, yielding a critical Damköhler number, i.e., the ratio of the expansion time scale to the homogeneous ignition time scale, given by (1−1)E0/RT−1/n, where n is the power law exponent of the self-similar expansion p = t−n. The critical ignition criterion, which is valid in the asymptotic limit n(1−1)E0/RT−1, was found in excellent agreement with numerical results. The applicability of the results obtained are discussed for ignition in rapidly expanding flows which occur behind decaying shock waves, as encountered in problems of detonation initiation by a Taylor-Sedov blast wave, and reacting jet start-up, and for reactions in steady hypersonic flows around projectiles.

1The present work was sponsored by an NSERC Discovery grant to MIR and by the NSERC Hydrogen Canada (H2CAN) Strategic Research Network.

4:14PM LU.00004 Effects of turbulence on syngas ignition in rapid compression machines, MATTHIAS IHME, ASKO SOIMAKALLIO, University of Michigan — Comparisons of ignition delays between predictions and measurements showed considerable differences for high-pressure/low-temperature syngas mixtures. Although effects of reaction-chemistry and large-scale hydrodynamic mixing have been identified as potential sources for these discrepancies, the significance of turbulence and turbulence/chemistry interaction has not be quantified. To address this issue, a theoretical model has been developed in which rapid-distortion theory and a Lagrangian Fokker Planck model have been combined to model turbulence amplification and autoignition in rapid compression machines (RCMs). The model was applied to a realistic RCM-configuration, and parametric studies were performed. From this study, a Damköhler criterion was derived to quantify the sensitivity of the induction chemistry to turbulence.
The spatial distribution of charged species in a methane-air counterflow diffusion flame is simulated with a detailed ion chemistry. The electric field induced by the distribution of charged species is calculated and compared to that obtained invoking the ambipolar diffusion assumption. The two calculations showed identical profiles for charged species and electric field. The profiles of ion mole fractions show two peaks: one near the maximum temperature and a second peak on the oxidizer side. The major ions near the maximum temperature are electron, $e^{-}$, and $H^+$, $O^+$, and $H_2O^+$. These profiles are quite different from those adopting a simplified three-step mechanism based solely on $E$, $CHO^-$, and $H_3O^+$, which shows only a single peak. Reaction pathway analyses showed that near the flame region, the proton is transferred by the path of $CHO^-$ to $H_2O^+$ as $C_2H_5OH$ form $CHO^-$ in a circulating manner. In the second peak, $CHO_2^-$ is produced through the pathway of $E^-\rightarrow O^-\rightarrow OH^-\rightarrow CHO_2^-$. The sensitivity of the charged species profiles to transport properties is investigated, and it is found that the variation of charged species profiles near peak temperature is relatively small, while on the oxidizer side, it is quite sensitive to transport properties.

This work was supported by CCRC KAUST.

A numerical study is conducted to evaluate the performance and optimal operating conditions of fluidized-bed reactors for fast pyrolysis of biomass to bio-oil. A comprehensive CFD model, coupling a pyrolysis kinetic model with a detailed hydrodynamics model, is developed. A lumped kinetic model is applied to describe the pyrolysis of biomass particles. Variable particle porosity is used to account for the evolution of particle physical properties. The kinetic scheme includes primary decomposition and secondary cracking of tar. Biomass is composed of reference components: cellulose, hemicellulose, and lignin. Products are categorized into groups: gaseous, tar vapor, and solid char. The particle kinetic processes and their interaction with the reactive gas phase are modeled with a multi-fluid model derived from the kinetic theory of granular flow. The gas, sand and biomass constitute three continuum phases coupled by the interphase source terms. The model is applied to investigate the effect of operating conditions on the tar yield in a fluidized-bed reactor. The influence of various parameters on tar yield, including operating temperature and others are investigated. Predicted optimal conditions for tar yield and scale-up of the reactor are discussed.

The evolution of soot in turbulent reacting flows is driven by the small-scale interactions between turbulence, chemistry, and soot. Due to an infinite Schmidt number, soot is confined to thin structures which are stretched by small-scale interactions. Soot particles are used to examine the nature of acoustically generated forces. The present experimental configuration provides a useful test bed for the evaluation of the response of different burning fuels to an acoustically resonant environment, including conditions leading to flame extinction, and can be used to explore a range of condensed phase combustion processes during such acoustic coupling.

The sensitivity of the charged species profiles to transport properties is investigated, and it is found that the variation of charged species profiles near peak temperature is relatively small, while on the oxidizer side, it is quite sensitive to transport properties.

The sensitivity of the charged species profiles to transport properties is investigated, and it is found that the variation of charged species profiles near peak temperature is relatively small, while on the oxidizer side, it is quite sensitive to transport properties.
Comparison is also made to previous RANS/PDF simulations. Eddy structures which have large scale in space and streamwise rotating axis are produced along the outer edge of OH layer in burnt gas. These streamwise eddies are induced by velocity difference due to strong expansion of the burnt gas. Although combustion condition of the present DNS is classified into corrugated flamelets regime, unburnt mixture islands frequently appear behind the main flame. The creation of these islands is closely related to the fine scale eddies in the unburnt turbulence and the separated unburnt mixture is consumed rapidly by the heating from surrounding burnt gas.

4:14PM LV.00004 Mixture fraction and its Dissipation in a turbulent flame by using Krypton PLIF, Venkat Narayanawamy, The University of Texas at Austin, Austin, TX. ANDREA HSU, Combustion Research Facility, Sandia National Laboratory, Livermore, CA. NOEL CLEMENS, The University of Texas at Austin, Austin, TX. JONATHAN FRANK, Combustion Research Facility, Sandia National Laboratory, Livermore, CA. COLLABORATION — A new idea of analysing turbulent premixed flames was inspired by dissipation element analysis developed by Wang and Peters. Starting from every point on the flame surface in the direction of ascending and descending gradient of the underlying curvature field, while not leaving the flame surface itself, one ends in a local maximum, respectively minimum point. The surface decomposes into fragments from which trajectories reach the same minimum and maximum. These fragments are surface-filling and do not overlap. Their geometry is non-arbitrary, since defined by the flame surface only. The linear distance between maximum and minimum of each fragment defines a length scale and the difference of curvature a scalar difference. The surface area of each fragment can then be estimated using the linear length and curvature difference. Therefore, estimating the whole flame surface area results in summing the areas of all fragments. Knowing the distribution function of the linear distance and the curvature difference can lead to a fine scale model of the flame surface area and the turbulent flame speed. DNS of the extended level-set G-equation in homogeneous isotropic forced turbulence have been performed in a cubic domain of $2\pi$ side length and 1024^{3} grid points.

4:27PM LV.00005 PDF-Based Simulations of Nonpremixed Turbulent Jet Flames With Advanced Radiation Models, G. PAL, M.F. MODEST, University of California-Merced, A. GUPTA, D.C. HAWORTH, The Pennsylvania State University — Transported probability density function (PDF) methods have been adopted as the basis for simulating turbulent reacting flows in both Reynolds-averaged and spatially filtered (large-eddy simulation) contexts. Skeletal gas-phase chemical mechanisms have been implemented for oxidation and pollutant formation in hydrocarbon-air flames, and detailed soot models have been implemented using a method-of-moments for soot aerosol dynamics. Photon Monte Carlo and high-order spherical harmonics radiative transfer equation (RTE) solvers have been coupled with the PDF method to deal with participating-medium radiation/turbulence/radiation interactions. Line-by-line and advanced k-distribution methods have been developed for spectral radiation properties. The result is a comprehensive framework for simulating luminous and nonluminous turbulent flames that accurately captures complex turbulence/chemistry/soot/radiation interactions. Quantitative comparisons with experimental measurements have been made. Systematic parametric studies have been performed to explore the relative importance of the choice of RTE solver versus the spectral radiation model.

This research has been supported under NASA-NNX07AB40A.

4:40PM LV.00006 Large-eddy simulation/probability density function modeling of local extinction and re-ignition in Sandia Flame E, HAIFENG WANG, PAVEL POPOV, VARUN HIREMATH, STEVEN LANTZ, SHARADHA VISWANATHAN, STEPHEN POPE, Cornell University — A large-eddy simulation (LES)/probability density function (PDF) code is developed and applied to the study of local extinction and re-ignition in Sandia Flame E. The modified Curl mixing model is used to account for the sub-filter scalar mixing; the ARM1 mechanism is used for the chemical reaction; and the in-situ adaptive tabulation (ISAT) algorithm is used to accelerate the chemistry calculations. Calculations are performed on different grids to study the resolution requirement for this flame. Then, with sufficient grid resolution, full-scale LES/PDF calculations are performed to study the flame characteristics and the turbulence-chemistry interactions. Sensitivity to the mixing frequency model is explored in order to understand the behavior of sub-filter scalar mixing in the context of LES. The simulation results are compared to the experimental data to demonstrate the capability of the code. Comparison is also made to previous RANS/PDF simulations.
A wetting liquid put into contact with a thin vertical tube rises spontaneously in it, reaching a final height of:

\[ \frac{h}{r} = 2 \left( \frac{\gamma}{\rho g \eta a} \right)^{1/2} \cos \theta \]

where \( r \) is the radius of the tube, \( a = \sqrt{a} \) is the capillary length, \( \gamma \) is the surface tension, \( \rho \) is the liquid density, \( g \) is gravity, \( \eta \) is the liquid viscosity, and \( \theta \) is the contact angle.

The front of the liquid follows Washburn's law:

\[ z = \frac{h_e}{\sqrt{2 g r c \eta}} \]

where \( h_e \) is given by Jurin's law:

\[ h_e = \frac{2 \gamma}{\eta a} \cos \theta \]

This works for all systems having a “closed” geometry, while for scaling length, provided this scaling length is smaller than \( a \).

We use systems of “open” geometry, without scaling length, typically wedges with different geometries and show both experimentally and theoretically that the meniscus rises following the universal law:

\[ \frac{h}{r} \sim \left( \frac{\eta a}{\rho g} \right)^{1/4} \]

This differs from the case of “closed” geometry because it rises indefinitely and with a different dynamic. It is universal in the sense that it does not depend on the special geometry of the wedge.

4:53PM LV.00007 Lagrangian and Eulerian PDF Simulations of Nonpremixed Turbulent Flames with Moderate-to-Strong Turbulence/Chemistry Interactions\(^1\). J. JAISHREE, D.C. HAWORTH, The Pennsylvania State University — Transported probability density function (PDF) methods offer compelling advantages for modeling chemically reacting turbulent flows, and Lagrangian particle-based Monte Carlo algorithms have become the predominant method for solving modeled PDF transport equations. Significant progress has been made in Lagrangian particle methods to facilitate their implementation into conventional Eulerian computational fluid dynamics (CFD) codes. Still, it would be desirable to realize the advantages of PDF methods using more conventional numerical algorithms and/or at lower computational cost. Toward these ends, Eulerian field methods have been proposed as alternatives to particle-based methods for solving modeled PDF transport equations. Here we apply Lagrangian particle and Eulerian field (both stochastic and deterministic) PDF methods to a hierarchy of three pilot methane-air nonpremixed turbulent jet flames where turbulence/chemistry interactions become progressively more important. Both accuracy and computational efficiency are assessed. Based on these results, we provide recommendations on how to apply each method most effectively, and we identify outstanding issues requiring future research.

\(^1\)This research has been supported by CD-adapco.

5:06PM LV.00008 Simulations of Turbulent Spray Combustion in a Constant-Volume Chamber for Diesel-Engine-Like Conditions. H. ZHANG, D.C. HAWORTH, The Pennsylvania State University — In-cylinder aero-thermal-chemical processes in piston engines are rich and complex, and modern engines are already at a high level of refinement. Further increases in performance, reductions in fuel consumption and emissions, and accommodation of nontraditional fuels will require the effective use of high-spatial-and-temporal-resolution optical diagnostics and numerical simulations. In this research, computational fluid dynamics tools are being developed to explore the influences of fuel properties on autoignition, combustion, and pollutant emissions in compression-ignition engines. The modeling includes a transported probability density function method to account for turbulent fluctuations in composition and temperature, detailed soot models with a method of moments for soot aerosol dynamics, a stochastic photon Monte Carlo method for participating-medium radiation heat transfer, and line-by-line spectral properties for mixtures of molecular gases and soot. The models are applied to a constant-volume spray combustion bomb where measurements are available for a range of thermochemical conditions and for a variety of fuels. Parametric studies of the influences of key physical and numerical parameters are performed to determine sensitivities and to establish best practices to be carried forward into subsequent modeling studies of real engines.

5:19PM LV.00009 Mechanism for AC Electric Field Deflection of Flames. MICHAEL CHEMAMA, KYLE BISHOP, LUDOVICO CADEMARTIRI, MICHAEL P. BRENNER, GEORGES M. WHITESIDES, Harvard University — Effects of electric fields on flames have been observed and studied since the 19th century. It is well known that the presence of an electric field can modify the shape of a burner or candle-like diffusion flame. Most experimental observations and theoretical analyses focused on DC fields. Recent experiments show that a flame can also be bent and even put out by an AC field. To explain how a zero time average cause can give rise to a net effect on the flame we develop a perturbation theory of the combustion equations modified to allow for the presence of the field and completed by Maxwell’s equation. Theoretical and numerical analyses of the equations indeed show that the AC field creates a force whose magnitude is comparable to gravity for high enough fields (1e5 V/m). The dependency of this critical field on the frequency and the effect on the flame shape are also obtained and compared to experimental results.

5:32PM LV.00010 Wildfire simulation using a chemically-reacting plume in a crossflow. ROBERT BREIDENTHAL, University of Washington, TRAVIS ALVARADO\(^1\), University of Texas, BRIAN POTTER, Pacific Wildland Fire Sciences Laboratory — Water tunnel experiments reveal the flame length of a chemically-reacting plume in a crossflow. Salt water containing a pH indicator and a base is slowly injected from above into the test section of a water tunnel containing an acidic solution. The flame length is measured optically as a function of the buoyancy flux, crossflow speed, and volume equivalence ratio of the chemical reaction. Based on earlier work of Broadwell with the transverse jet, a simple dilution model predicts the flame length of the transverse plume. The plume observations are in accord with the model. As with the jet, there is a minimum in the flame length of the plume at a transition between two self-similar regimes, corresponding to the formation of a pair of counter-rotating vortices at a certain crossflow speed. At the transition, there is a maximum in the entrainment and mixing rates. In an actual wildfire with variable winds, this transition may correspond to a dangerous condition for firefighters.

\(^1\)HACU Intern

Monday, November 22, 2010 3:35PM - 5:45PM — Session LW Surface Tension III Hyatt Regency Long Beach Regency C

3:35PM LW.00001 Capillary rise in wedges. ALEXANDRE PONOMARENKO, CHRISTOPHE CLANET, DAVID QUERE, ESPCI — A wetting liquid put into contact with a thin vertical tube rises spontaneously in it, reaching a final height of given by Jurin's law:

\[ \frac{h}{r} = 2 \left( \frac{\gamma}{\rho g \eta a} \right)^{1/2} \cos \theta \]

where \( r \) is the radius of the tube, \( a = \sqrt{a} \) is the capillary length, \( \gamma \) is the surface tension, \( \rho \) is the liquid density, \( g \) is gravity, \( \eta \) is the liquid viscosity, and \( \theta \) is the contact angle.

Also, when \( z \ll h_e \), where gravity can be neglected, the front of the liquid follows Washburn's law:

\[ z = \frac{h_e}{\sqrt{2 g r c \eta}} \]

where \( h_e \) is given by Jurin’s law:

\[ h_e = \frac{2 \gamma}{\eta a} \cos \theta \]

This works for all systems having a “closed” geometry, while for scaling length, provided this scaling length is smaller than \( a \).

We use systems of “open” geometry, without scaling length, typically wedges with different geometries and show both experimentally and theoretically that the meniscus rises following the universal law:

\[ \frac{h}{r} \sim \left( \frac{\eta a}{\rho g} \right)^{1/4} \]

This differs from the case of “closed” geometry because it rises indefinitely and with a different dynamic. It is universal in the sense that it does not depend on the special geometry of the wedge.

3:48PM LW.00002 Delayed Coalescence of Sessile Droplets with Different but Miscible Liquids. STEFAN KARPITSCHKA, HANS RIEGLER, Max Planck Institute of Colloids and Interfaces, D-14476 Potsdam, Germany — Due to capillary forces, two sessile droplets of miscible liquids will fuse when they contact each other. Usually the droplet fusion proceeds very fast, delayed mostly by viscous forces. However, quite unexpectedly, it was observed recently\(^2\) that the coalescence of sessile droplets of completely miscible liquids can be delayed up to minutes. After a first contact, the droplets remain separated by a thin liquid neck and push each other across the substrate before they finally merge. The delayed coalescence may be highly relevant for technical applications, for instance in the sector of semiconductor manufacturing, where controlled contact line displacement is a key technology. It is assumed that the coalescence is delayed by a Marangoni convection through the thin film connecting the drops. This suggests that the effect is quite common. A sharp transition from fast to delayed coalescence was found\(^3\) as the difference in surface tensions exceeded \( 4 \text{ mN/m} \), irrespective of other liquid properties like absolute surface tensions or viscosities. We present new experimental results addressing the dynamics of the liquid neck between the drops from which we can distinguish various coalescence modes.


A numerical method for variable surface tension effects in non-isothermal atomization with overset grids.

An experimental investigation of the theory of electrostatic deflections.

Ant tower.

Wet-dog shake.

Elastocapillary imbibition.

Capillary rafts and their destabilization.

Boiling phenomena of capillary flow in heat pipe applications.
A well-defined leading-edge vortex, which remains in a stable position, is attainable over a wide range of effective angle of attack, up to 75 degrees. These techniques of particle image velocimetry lead to patterns of vorticity in relation to the sectional streamline topology. Values of Reynolds number, attained by using eddy-current techniques with different pitching instability in the on-road test. The simplified vehicle modes were used so as to reproduce the characteristic flow structures above the trunk deck of the real vehicles measured in a wind-tunnel at the static case without oscillation. The forced sinusoidal pitching oscillation was imposed on the models and their pitching damping factors were evaluated through the phase-averaged pitching moment. Then flow structures in the wake of the models were extracted and its contribution to the damping mechanism was discussed. It was found that slight difference of the front and rear pillars' shape drastically affects the flow structures in the wake of the models, which enhance or restrain the vehicles' pitching instability.

A computational study of golfball aerodynamics: effects of rotation. An efficient finite-difference Navier-Stokes solver is used to carry out a series of simulations of a spinning golfball at three distinct flow regimes: subcritical, critical and super-critical. The golfball is treated using an embedded boundary formulation, where the velocity near the surface is locally reconstructed to satisfy the proper boundary conditions. All scales down to separation or installing small components either to reduce underbody flow or to mitigate induced drag. In this study, we examine the effect that dimples have upon the aerodynamics of a simplified vehicle. Reynolds-averaged Navier-Stokes simulations are performed on a full-scale Ahmed body at a Reynolds number of 9.5e6 based upon the vehicle length. The dimples, which have a uniform diameter of 0.1 m and a dimple depth-to-diameter ratio of 0.14, are distributed across the vehicle surface. The results of the simulations demonstrate that the dimples modify both the recirculation zone and the strength and location of the counter-rotating vortex pair in the vehicle wake. Although an increase in base pressure can occur for a dimpled configuration, the net drag change is sensitive to both the number and placement of the dimples on the vehicle body.

Aerodynamic shape optimization of Airfoils in 2-D incompressible flow. On a stand-alone computer, a run takes about an hour to obtain a converged solution. The airfoil geometry was generated using two Bezier curves; one to represent the thickness and the other the camber curve. The coefficient of lift and drag was computed using potential velocity distribution obtained from panel code, and boundary layer transition prediction code was used to predict the location of onset of transition. The objective function of a particular design is evaluated as the weighted-average of aerodynamic characteristics at various angles of attacks. Optimization was carried out for several objective functions and the airfoil designs obtained were analyzed.

Simulation of a Plunging Airfoil with a Flexible Tail. The airfoil geometry was generated using two Bezier curves; one to represent the thickness and the other the camber curve. The coefficient of lift and drag was computed using potential velocity distribution obtained from panel code, and boundary layer transition prediction code was used to predict the location of onset of transition. The objective function of a particular design is evaluated as the weighted-average of aerodynamic characteristics at various angles of attacks. Optimization was carried out for several objective functions and the airfoil designs obtained were analyzed.

A Panel-Particle Method for Studies of Maneuvering Aircraft Formations. An efficient finite-difference Navier-Stokes solver is used to carry out a series of simulations of a spinning golfball at three distinct flow regimes: subcritical, critical and super-critical. The golfball is treated using an embedded boundary formulation, where the velocity near the surface is locally reconstructed to satisfy the proper boundary conditions. All scales down to separation or installing small components either to reduce underbody flow or to mitigate induced drag. In this study, we examine the effect that dimples have upon the aerodynamics of a simplified vehicle. Reynolds-averaged Navier-Stokes simulations are performed on a full-scale Ahmed body at a Reynolds number of 9.5e6 based upon the vehicle length. The dimples, which have a uniform diameter of 0.1 m and a dimple depth-to-diameter ratio of 0.14, are distributed across the vehicle surface. The results of the simulations demonstrate that the dimples modify both the recirculation zone and the strength and location of the counter-rotating vortex pair in the vehicle wake. Although an increase in base pressure can occur for a dimpled configuration, the net drag change is sensitive to both the number and placement of the dimples on the vehicle body.

Aerodynamics of a Dimpled Vehicle. An efficient finite-difference Navier-Stokes solver is used to carry out a series of simulations of a spinning golfball at three distinct flow regimes: subcritical, critical and super-critical. The golfball is treated using an embedded boundary formulation, where the velocity near the surface is locally reconstructed to satisfy the proper boundary conditions. All scales down to separation or installing small components either to reduce underbody flow or to mitigate induced drag. In this study, we examine the effect that dimples have upon the aerodynamics of a simplified vehicle. Reynolds-averaged Navier-Stokes simulations are performed on a full-scale Ahmed body at a Reynolds number of 9.5e6 based upon the vehicle length. The dimples, which have a uniform diameter of 0.1 m and a dimple depth-to-diameter ratio of 0.14, are distributed across the vehicle surface. The results of the simulations demonstrate that the dimples modify both the recirculation zone and the strength and location of the counter-rotating vortex pair in the vehicle wake. Although an increase in base pressure can occur for a dimpled configuration, the net drag change is sensitive to both the number and placement of the dimples on the vehicle body.

4:01PM LX.00003 Aerodynamics of a Dimpled Vehicle, JASON ORTEGA, KAMBIZ SALARI, Lawrence Livermore National Laboratory — Automobiles consume approximately two billion barrels of fuel each year throughout the United States. A significant portion of this fuel is used to overcome aerodynamic drag at highway speeds. As a result, even small improvements made to the aerodynamics of automobiles can result in sizeable fuel savings. Since the shape of a vehicle is often dictated by design, economics, and function, aerodynamic improvements by means of obvious body streamlining are not always possible. However, minor modifications can be made to the vehicle, such as changing the behavior of the boundary layer to delay flow separation or installing small components either to reduce underbody flow or to mitigate induced drag. These modifications have been made on controlling and optimizing such formations, these analyses have traditionally been restricted to quasi-steady aerodynamic models with flat wakes. Such models, though insightful, fail to capture essential wake dynamics during maneuvers. Here, a hybrid panel-particle method is presented to introduce unsteady wake effects in the study of formation flight systems. A fast-multipole algorithm is used for numerical speed-up. This approach is less computationally expensive than high-fidelity CFD, but is still able to capture essential wake physics lacking in quasi-steady approaches. Moreover, the representation of the wake as vortex particles allows for studies of wake-body interactions to be handled with greater ease. This presentation will introduce the method and demonstrate its merits in simulating single and multiple aircraft undergoing maneuvers.

4:40PM LX.00006 Simulation of a Plunging Airfoil with a Flexible Tail, ALAN LAI, FENG LIU, University of California, Irvine — The fluid motion of an airfoil with a flexible tail is simulated using an unsteady panel method with diffusive wake modeling. The fluid simulation is coupled with a CSD solver to simulate the deflection of the flexible tail due to both inertial and aerodynamic loads. The modal equations were used to calculate the structural deformation under an aerodynamic load. Computations with varying stiffness coefficient and reduced frequencies were performed to produce a performance map of a plunging airfoil with a flexible tail. The results showed a range of reduced frequencies and tail stiffness that increased the thrust produced by the plunging motion by as much as 45% when compared with the same airfoil that has a rigid tail. The propulsive efficiency with a flexible tail increased slightly as well. The upper limit of the thrust enhancement is bounded by the first natural frequency of the flexible tail. A stiffer tail is shown to be most beneficial, but the minimum reduced frequency where thrust is improved increases with increasing stiffness. Spectral analysis of the unsteady forces and wake velocities showed that the increase in thrust can be directly attributed to the effect the flexible tail has on the wake vortices.

5:06PM LX.00008 Flow Structure on a Rotating Plate, CEM OZEN, DONALD ROCKWELL, Lehigh University — The flow structure on a rotating (revolving) plate with an aspect ratio of one is considered for the case of steady plate rotation, well after the transient startup. Techniques of particle image velocimetry lead to patterns of vorticity in relation to the sectional streamline topology. Values of Reynolds number, attained by variation of the angular velocity of the plate, range from approximately 3,000 to 13,000. The observed patterns are relatively insensitive to Reynolds number. A well-defined leading-edge vortex, which remains in a stable mode, is attainable over a wide range of effective angle of attack, up to 75 degrees. These quasi-two-dimensional features are intimately related to three-dimensional characteristics of the flow, which involves spanwise-oriented patterns of velocity that vary along the chord of the plate, as well as distinctive patterns in the vicinity of the root and the tip of the rotating plate. The flow structure on the corresponding plate undergoing steady, purely translational motion is directly compared with that along the rotating plate. The flow pattern is fundamentally altered in absence of rotation, with dominance of large-scale stall.
An investigation of the influence of freestream turbulence on the laminar separation bubble of an SD7003 airfoil at low Reynolds number. Freestream turbulence acts as an excitation source that can influence the evolution of the boundary layer and the separated shear layer. The influence of freestream turbulence is studied by deliberately modifying the freestream turbulence using a turbulence-generating grid upstream of the airfoil. Multiline single-component Molecular Tagging Velocimetry (MTV) with its high resolution near-wall measurement capability (approximately 10 times better cross-stream resolution than recent PIV studies) is utilized. Results with/without the grid are compared with computations and other experiments in different facilities.

Supported by AFRL through the Michigan/AFRL Collaborative Center in Aeronautical Sciences.

Monday, November 22, 2010 3:35PM - 5:45PM  
Session LY Viscous Flows  
Hyatt Regency Long Beach Regency E

3:35PM LY.00001 Exchange flow of two immiscible fluids and the principle of maximum flux  
JAMES HANNA, DAVID OLSON, ALAN KATZ, AHMED NAGUIB, MANOOCHEHR KOOCHESFARHANI, Michigan State University, DONALD RIZZETTA, MIGUEL VISBAL, Air Force Research Laboratory, Wright-Patterson Air Force Base — There is considerable discrepancy in the literature regarding the location of separation and reattachment points on the steady SD7003 airfoil obtained in different experimental and computational studies. Among several factors that could lead to this discrepancy in experiments, the facility’s freestream turbulence level is believed to be important. Freestream turbulence acts as an excitation source that can influence the evolution of the boundary layer and the separated shear layer. The influence of freestream turbulence is studied by deliberately modifying the freestream turbulence using a turbulence-generating grid upstream of the airfoil. Multiline single-component Molecular Tagging Velocimetry (MTV) with its high resolution near-wall measurement capability (approximately 10 times better cross-stream resolution than recent PIV studies) is utilized. Results with/without the grid are compared with computations and other experiments in different facilities.

3:48PM LY.00002 A New Theory of Oscillating Flows  
Vladimir Vladimirov, Mathematics, University of York, UK — A new theory of viscous oscillatory flows has been developed. Our theory represents an adaptation of the Vishik-Lyusternik approach combined with the two-timing and averaging methods. We consider the high Re viscous incompressible flows driven by a vibrating boundary for the simple geometry of a half-space.

From the physical viewpoint the considered boundary conditions may be seen as the tangential vibrations of material points of a plane stretchable membrane. The main result is the obtaining of the general, global, and uniformly valid asymptotic solutions of the Navier-Stokes equations. These solutions satisfy general oscillating boundary conditions and three different settings of the scaling parameters (that correspond to the strong, moderate, and weak nonlinearities). We have derived that the streaming part of any solution is governed by either the full Navier-Stokes equations or Stokes’ equations (both with the unit Re) as well as by the precisely derived effective boundary conditions. The examples of the spatially periodic vibrations of the boundary and the angular torsional vibrations of an infinite rigid disc have been considered. In the sharp contrast to all previous theories of oscillating flows (see e.g. Batchelor’s “Introduction to fluid dynamics,” formula 5.13.15) our solutions do not deal with any secular (infinitely growing with the inner normal coordinate) terms. This new approach may be seen as a revolutionary step in the field, since for the very first time it does not use the asymptotic matching procedures and the boundary layer theories.

4:01PM LY.00003 The asymptotic structure of a slender coiling fluid thread  
MAURICE BLOUNT, JOHN LISTER, University of Cambridge — The buckling of a viscous thread as it falls through air onto a stationary surface is a well-known breakfast-time phenomenon which exhibits a rich variety of dynamical regimes [1]. Since the bending resistance of a slender thread is small, bending motion is largely confined to a short region of coiling near the surface. If the height of fall is large enough, then the thread above the coiling region forms a ‘tail’ that falls nearly vertically under gravity but is deflected slightly due to forces exerted on it by the coil. Although it is possible to use force balances in the coil to estimate scalings for the coiling frequency, we analyse the solution structure of the entire thread in the asymptotic limit of a very slender thread and thereby include the dynamic interaction between the coil and the tail. Quantitative predictions of the coiling frequency are obtained which demonstrate the existence of leading-order corrections to scalings previously derived. In particular, we show that in the regime where the deflection of the tail is governed by a balance between centrifugal acceleration, hoop stress and gravity, the tail behaves as a flexible circular pendulum that is forced by bending stress exerted by the coil. The amplitude of the response is calculated and the previously observed resonance when the coiling frequency coincides with one of the eigenfrequencies of a free flexible pendulum is thereby explained. [1] N.M. Ribe et al., J. Fluid Mech. 555, 275-297.

4:14PM LY.00004 The shape of an elastic filament in a two-dimensional corner flow  
LAURA GUGLIELMINI, Princeton University, NICOLAS AUTRUSSE, Institute Superior de l’Aeronautique et de l’Espace, SIGOLENE LECUYER, Harvard University, ROBERTO RUSCONI, Massachusetts Institute of Technology, HOWARD STONE, Princeton University — The deformation of a flexible filament held fixed at one end in a nonuniform viscous flow with curved streamlines is considered, with a focus on the filament dynamics and steady-state shape. Our motivation arises from recent microfluidic experiments on biofilm formation: in a channel with bends, thread-like structures, or streamers, were observed, attached to the side walls downstream of each corner and connecting consecutive corners while floating in the channel middle plane (Rusconi et al., J. Roy. Soc. Interface 2010). We discuss the time evolution and final shape of the filament in different corner geometries as a function of a non-dimensional elasticity parameter that compares viscous and elastic effects. Since the filament develops tension, when the flow has curved streamlines the filament does not align with the flow, but rather it crosses the streamlines, in contrast with the behavior observed for rectilinear flows. We also discuss the buckling instabilities that occur when the filament undergoes compression for the specific case of a stagnation point flow near a wall.

4:27PM LY.00005 Shape dynamics of a thin loop sedimenting in a viscous fluid  
JAMES HANNA, CHRISTIAN SANTANGELO, Department of Physics, UMass-Amherst — Thin elastic filaments and chains in viscous fluids are idealizations of biological and polymeric systems. We consider the non-local shape evolution due to hydrodynamic self-interaction, of a chain-like, locally inextensible loop settling under gravity in the creeping flow regime. We find that the rigid translation of a circle along its gravitationally-aligned axis is unstable. A recirculating blob and tail arrangement is explored as a possible attracting state of a long chain.

4:40PM LY.00006 Slender soft-magnetic body in highly viscous flows  
JAMES MARTINDALE, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, LEANDRA VICCI, LONGHUA ZHAO, The University of North Carolina at Chapel Hill, NSF RTG FLUIDS GROUP TEAM — For a tilted soft-magnetic slender body in a highly viscous fluid whose motion is driven by a prescribed background rotating magnetic field, the interaction between hydrodynamic and magnetic forces must be understood in order to predict the combined motion of a rod in silicon oil. Such a system arises in an experiment emulating primary cilia-driven fluid flows in developing embryos. Using classical slender body theory, the magnetic contribution to this dynamical system results in a system of equations for the torque balance. Further, analysis of body geometry such as fixed curvature will be explored, and their influence on the fluid motion illustrated.

Funded by NSF RTG.
bottom plates and of suspended, thermally active particles on the stability threshold is also investigated. Solving in closed form which differs by less than 1% from the exact result. Using the same procedure, the effect of finite thermal conductivities of the top and bottom plates of blood flow in the human abdominal aorta.

TOBIAS M. SCHNEIDER, MICHAEL P. BRENNER, School of Engineering and Applied Sciences, Harvard University — Electrical properties of semiconductors are mainly controlled by the concentration of dopants. While the highest dopant concentrations reachable in most traditional doping methods are limited by the equilibrium solubility of the dopant in the pure semiconductor material, much higher concentrations are observed after femtosecond laser treatment of silicon in a sulfur containing atmosphere. We propose a mechanism underlying this observation and show that the Laser induced melting dynamics of the silicon surface in combination with dopant diffusion alone can give rise to the observed high doping concentrations. Modeling the re-solidification of the binary silicon-sulfur mixture in a semi-infinite domain allows to predict the dependence of doping concentrations on depth and provides a method to control the shape of concentration profiles and thereby electrical properties of the semiconductor material. Controlling these properties in a range not accessible to traditional doping methods provides new avenues for optimizing the efficiency of photovoltaic cells.

DOMINIC VELLA, HERBERT HUPPERT, University of Cambridge — Axisymmetric similarity solutions for viscous gravity currents spreading on a uniform plane are well-known. The addition of regions of leakage destroys both the symmetry and the self-similarity by introduction of preferred directions and length scales. We examine the effect of such leaks on the asymptotic long-time behaviour of viscous and porous gravity currents in a variety of geometries, showing how novel self-similar structures can emerge in the far-field, fed by quasi-steady near-field solutions that are nonlinear analogues of problems in electrostatics. Matching the two allows predictions of the asymptotic rate of spread and increase in volume of the current. The problems are motivated by the proposed geological sequestration of CO$_2$ and bear on the time scales over which CO$_2$ may be stored in saline aquifers that have an imperfect seal in the cap rock.
4:14PM LZ.00004 Comparison of Linear Stability and 3-D Time Integration for Predicting Instabilities in a Thermocapillary Driven Liquid Bridge with Magnetic Stabilization\textsuperscript{1}, KENNETH DAVIS, YUE HUANG, BRENT HOUCHENS, Rice University — Flow in a cylindrical liquid bridge is driven by thermocapillary effects arising from a temperature gradient applied on the free surface. When the temperature difference is small and axisymmetric, the base flow will also be axisymmetric. However, as the temperature difference increases the flow becomes susceptible to three-dimensional instabilities, the first of which is either stationary or periodic depending on the Prandtl number of the liquid bridge. Instabilities predicted by linear stability theory are compared with those found using three-dimensional time integrations for low Prandtl number liquid bridges. Comparisons are drawn between spectral collocation and spectral element simulations, in terms of accuracy and computational efficiency. The impact of stabilizing the base state by applying a steady, axial magnetic field is also investigated.

\textsuperscript{1}This work was supported by the U.S. Air Force Office of Scientific Research

4:27PM LZ.00005 Convection onset in colloidal suspensions of particles, LAYACHI HADJI, University of Alabama — A particular medium model is used to investigate the onset of Rayleigh-Bénard convection in a colloidal suspension of inert solid particles. The model accounts for the effects of thermophoresis, sedimentation and Brownian diffusion. Depending on the size of the particles, the problem has up to four time scales. These are due to thermal diffusion, particle diffusion, particle migration due to thermophoresis, and sedimentation. The ratios of these time scales lead to the emergence of three parameters, one of which is the Lewis number \( \tau \). The smallness of the latter makes the differential eigenvalue system governing convection onset singular. The other two are the density number \( \Gamma \) and the dimensionless migration velocity \( \beta \). For a given experimental set-up, \( \beta \) can be viewed as a function of the particles’ radius. A combination of asymptotics and numerical computations is used to capture the effect of the resulting thin particle concentration boundary layers on the leading order threshold values of the Rayleigh number \( R_c \). Results, which are depicted as function of \( \Gamma \) and \( \beta \), reveal a non-monotonic dependence of \( R_c \) on \( \beta \). The curve \( R_c(\beta) \) is bimodal and it exhibits a maximum \( R_c^{*} \), the value of which increases very sharply with \( \Gamma \) while the critical wavelength decreases, at \( \beta \) values that correspond to nano sized particles. This implies that parameter controllers can be controlled so that the mixing of a small amount of nano size particles has a substantial stabilizing effect.

4:40PM LZ.00006 Magnetic Fields Applied to Paramagnetic Suspensions: The Hump-Jet Transition, SCOTT S.H. TSAI, Harvard University, Cambridge, MA, ZHENZHEN LI, ESPCI, Paris, France, PILNAM KIM, HOWARD A. STONE, Princeton University, Princeton, NJ — When a suspension of paramagnetic beads is in a sufficiently strong magnetic field gradient, a jet forms. Based on this approach, we report a technique for depositing an aggregate of paramagnetic beads on a substrate. Our setup is similar to the classical electrohydrodynamic jet setup originally used by Zeleny (1917), Wilson and Taylor (1925), who investigated the case of a single-phase liquid. In contrast, our system consists of a dilute suspension of micron-size paramagnetic beads suspended in the fluid. In response to a weak magnetic field, all of the beads collect at the almost planar interface, which then deforms modestly as the field strength is increased to form a hump. Above a critical field strength, the hump where the beads have collected goes unstable to form a jet. We use high-speed videos to study the system’s hump-jet transition. We also propose an analytical scaling model that predicts the critical conditions for the transition by the balance of magnetic and capillary forces acting on the aggregate of beads.

4:53PM LZ.00007 Linear stability analysis of capillary instabilities for concentric cylindrical shells\textsuperscript{1}, XIANGDONG LIANG, DAOSHENG DENG, Massachusetts Institute of Technology, JEAN-CHRISTOPHE NAVE, McGill University, STEVEN G. JOHNSON, Massachusetts Institute of Technology — We present a linear stability analysis of capillary instabilities in concentric cylindrical flows of \( \lambda \) fluids with arbitrary viscosities, thicknesses, and surface tensions. This generalizes previous work by Tomotika (\( N = 2 \)) and Stone & Brenner (\( N = 3 \), equal viscosities). We briefly explain the derivation, consider interesting limiting cases for \( N = 3 \) and \( N \to \infty \), and predict a phenomenon of competing breakup lengthscales in a 3-fluid system that we demonstrate with full 3d calculations.

\textsuperscript{1}This work was supported in part by the MRSEC Program of the National Science Foundation under award number DMR-0819762.

5:06PM LZ.00008 Generalized Rayleigh-Taylor and Richtmyer-Meshkov instabilities in particle-seeded flow\textsuperscript{1}, PETER VOROBIEFF, JOSEPH CONROY, MICHAEL ANDERSON, ROSS WHITE, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas-Brownsville — We describe a hydrodynamic instability analogous to Rayleigh-Taylor (RT) and Richtmyer-Meshkov (RM) instabilities in gravity-driven or impulsively-accelerated two-phase flows where the seeding density of the second phase (and the resulting average density) is initially non-uniform. The forcing causes the second phase (in our experiments, submicron-sized droplets in gas) to move with respect to the embedding medium. With sufficient seeding concentration, this leads to entrainment of the embedding phase. The resulting movement is similar to the movement that would evolve in a mixing flow with no second phase seeding, but with non-uniform density (e.g., a mixture of lighter and heavier gases), where RT and RM instabilities develop in the case of gravity-induced and impulsive acceleration respectively.

\textsuperscript{1}Special thanks to S. “Bala” Balachandar (U. of Florida) for inspiring this work. This research is funded by NNSA through DOE Grant DE-PS52-08NA28920 and by DTRA awards HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

5:19PM LZ.00009 Mixing two miscible liquids with Faraday instabiliy , FARZAM ZOUESHTIAGH, Institut d’Electronique, de Microélectronique et de Nanotechnologie (IEMN) UMR CNRS 8520, Avenue Poincaré, 59652 Villeneuve d’Ascq, France, SAKIR AMIROUDINE, Laboratoire TREFLE, Esplanade des Arts et Métiers, F-33405 Talence, France, RANGA NARAYANAN, University of Florida, FL 32611 Gainesville, USA — Generation of waves near the interface of one or two liquid layers that is subjected to vertical vibration is known as the Faraday instability. This instability occurs on account of a resonance that is set up when there is a tuning of the imposed frequency with the natural frequency of the free surface which possesses surface potential energy. Now if the free surface was removed by completely confining the container then no such instability could occur unless potential energy was introduced in some other way, say via density gradients. In this regard, we have recently seen experimentally and numerically that Faraday type of instability can also occur between two miscible liquids with different densities [1]. Here, we report on experimental and numerical study of Faraday instability used as a mixing tool. In particular, we characterize the mixing efficiency by the instability by measuring the size of the volume where the two liquids were fully mixed under different external vibration parameters.


5:32PM LZ.00010 Buoyancy-driven instabilities of miscible two-layer stratifications , P.M.J. TREVELYAN, C. ALMARCHA, A. DE WIT, NLPC, Université Libre de Bruxelles, Belgium — Buoyancy-driven instabilities of a horizontal interface between two miscible solutions in the gravity field are studied for porous media by a theoretical approach. Beyond the classical Rayleigh-Taylor and double-diffusive instabilities that can affect such a two-layer stratification right at the initial time of contact, diffusive-layer convection as well as delayed-double diffusive instabilities can set in at later time when differential diffusion effects act upon the evolving density profile starting from a step-function initial condition between the two miscible solutions. The conditions for these instabilities to occur can therefore not be obtained using linear base state profiles but can be computed only by considering time evolving base state profiles. To do so, we perform a linear stability analysis based on a quasi steady state approximation as well as nonlinear simulations of a diffusion-convection model. We classify and analyze all possible buoyancy-driven instabilities of a stratification of a solution of A on top of a miscible one of B as a function of a buoyancy number \( R \) quantifying the ratio of the relative contribution of B and A to the density and of \( \delta \) the ratio of diffusion coefficients of these two species.
derive a probability density function (PDF) of local length scales rather than a single scale. By connecting a local dissipation scale prescribed using mean dissipation rate, whereas dissipation is spatially intermittent. It therefore seems natural to instead describe dissipation using a continuum approach.

The first order terms lead to alternate functional equations, arising from ratios of two successive derivatives of the functional equations, each of which admits a constant is shown to be zero. In the present work, two terms outer layer expansion is considered where leading term scales with free steam velocity and first order with friction velocity. The leading term turns out to be a non-linear wake type equation through application of Iakazon-Millikan- Kolmogorov hypothesis. The first order terms lead to alternate functional equations, arising from ratios of two successive derivatives of the functional equations, each of which admits a constant is shown to be zero. In the present work, two terms outer layer expansion is considered where leading term scales with free steam velocity and first order with friction velocity. The leading term turns out to be a non-linear wake type equation through application of Iakazon-Millikan- Kolmogorov hypothesis.

The viscous sublayer of wall bounded turbulent flows is a thin region, usually assumed to stretch out to about 5 viscous length units, where the mean velocity distribution is close to linear. Its thickness is typical of the order of one percent or less of the boundary layer thickness or channel height. Despite this fact its importance for the flow in the boundary layer cannot be overstated since the mean shear stress at the wall determines the velocity scale of the Reynolds stresses and hence the velocity scale of the turbulence itself. In this presentation we show how the variation of the flow statistics within the viscous sublayer can be understood from a simple analysis of the instantaneous velocity profile. Special emphasis is put on the near self-similarity of the probability density distribution (pdf) of the streamwise velocity in the viscous sublayer. We also describe how the pdf of the fluctuating streamwise velocity measured using hot-wire anemometry can be used to determine the wall position and the friction velocity despite the fact that such measurements are contaminated by interference effects close to the wall. We illustrate this analysis both with DNS results from turbulent boundary layers and channel flows as well as from experiments in turbulent boundary layers.

Supported by ONR and DFG.

1 This work was financially supported by the US NSF through NSF/DMR grant 06-04477 and NSF/DMR grant 06-04455. T.T. acknowledges support from the Vietnam Education Foundation. P.C. acknowledges support from the Roscoe G. Jackson II Research Fellowship.

8:26AM MA.00003 Macroscopic effects of the spectral structure in turbulent flows1. T. TRAN, P. CHAKRABORTY, University of Illinois, N. GUTTENBERG, A. PRESCOTT, H. KELLAY, Centre de Physique Moléculaire Optique et Hertzienne (UMR 5798 CNRS), France, W. GOLDBURG, University of Pittsburgh, N. GOLDFELD, G. GIOIA, University of Illinois — There is a missing link between macroscopic properties of turbulent flows, such as the frictional drag of a wall-bounded flow, and the turbulent spectrum. To seek the missing link we carry out unprecedented experimental measurements of the frictional drag in turbulent soap-film flows over smooth walls. These flows are effectively two-dimensional, and we are able to create soap-film flows with the two types of turbulent spectrum that are theoretically possible in two dimensions: the "enstrophy cascade," for which the spectral exponent α = 3, and the "inverse energy cascade," for which the spectral exponent α = 5/3. We find that the functional relation between the frictional drag f and the Reynolds number Re depends on the spectral exponent: where α = 3, f ∼ Re−1/2; where α = 5/3, f ∼ Re−1/4. Each of these scalings may be predicted from the attendant value of α by using a recently proposed spectral theory of the frictional drag. In this theory the frictional drag of turbulent flows on smooth walls is predicted to be f ∼ Re(1−α)/(1+α).

8:39AM MA.00004 Mean velocity of a free fully-developed turbulent boundary layer . LEI XU, ZVI RUSAK, LUCIANO CASTILLO, Rensselaer Polytechnic Institute — A novel model equation for describing the profiles of the mean axial velocity of a free fully developed turbulent boundary layer over a smooth solid wall at zero pressure gradient is developed. The model uses the Reynolds-average equation for the mean axial speed and a modified Prandtl's mixing-length curve for the turbulent stress. This model is used to integrate the mean velocity profiles and compute the wall friction coefficient along the wall. The computed results of the velocity profiles and the friction coefficient show a remarkable agreement with much measured data and results from direct numerical computations for a wide range of Re° between 10 and 100 million, except for the transition region. Moreover, the present analysis demonstrates the four main regions that govern the flow and sheds new light on the structure of the boundary layer.

8:52AM MA.00005 The turbulent mean-velocity profile: it is all in the spectrum . GUSTAVO GIOIA, NICHOLAS GUTTENBERG, NIGEL GOLDFELD, PINAKI CHAKRABORTY, University of Illinois — It has long been surmised that the mean-velocity profile (MVP) of a pipe flow is closely related to the spectrum of turbulent energy. Here we perform a spectral analysis to identify the eddies that dominate the production of shear stress via momentum transfer. This analysis allows us to express the MVP as a functional of the spectrum. Each part of the MVP relates to a specific spectral range: the buffer layer to the dissipative range, the log layer to the inertial range, and the wake to the energetic range. The parameters of the spectrum set the thickness of the viscous layer, the amplitude of the buffer layer, and the amplitude of the wake.
9:18AM MA.00007 Relevant length scales in wall-bounded turbulence. SUBHAS VENAYAGAMOORTHY, LAKSHMI DASI, Colorado State University — The structure of wall-bounded turbulence is different from the case of isotropic/homogeneous turbulence due to the presence of local mean shear rate $S$. $S$ produces kinetic energy that have been classically assumed to occur at length scales of the order of the integral length scale. We show that there are three independent length scales that are regulated by $S$ interacting with kinetic energy $k$, viscosity $\nu$ and dissipation rate $\epsilon$. The first two length scales: $L_{S,k}$ based on $S$ and $k$; and $L_{S,\nu}$ based on $S$ and $\nu$; signify the upper and lower bounds of the scales that represent turbulence production respectively. The third length scale, $L_{S,\epsilon}$ which is based on $S$ and $\epsilon$, is an intermediate scale that signifies the beginning of overlap between the energy cascade process and the production range set by the first two scales. We also illustrate the fundamental and independent nature of these three length scales in that they set two important classical length scales of motion in wall-bounded turbulence, namely the large eddy length scale $L_{k,\nu}$ based on $k$ and $\nu$; the Kolmogorov length scale $\eta$; and a third new smallest length scale $L_{S,\epsilon}$ based on $k$ and $\nu$. Analysis of the variation of all six length scales using a large high-resolution DNS database of turbulent channel flow is provided with fresh insights into the dynamic characteristics that define the viscous sublayer, buffer layer, and the inertial regime.

9:31AM MA.00008 New insights into adverse pressure gradient boundary layers, WILLIAM K. GEORGE, MICHEL STANISLAS, JEAN-PHILIPPE LAVAL, Laboratoire de Mécanique de Lille, Univ Lille Nord de France — In a recent paper Shah et al. 2010 (Proc. of the WALLTURB Meeting, 2009), Lille, FR, Springer, in press) documented a number of adverse pressure gradient flows (APG’s), with and without wall curvature, where the turbulence intensity peak moved quite sharply away from the wall with increasing distance. They further suggested that this peak was triggered by the adverse pressure gradient and had its origin in an instability hidden in the turbulent boundary layer, developing soon after the change of sign of the pressure gradient. They then offered that this may explain the difficulties encountered up to now in finding a universal scaling for turbulent boundary layers. We build on these observations, and show that in fact there is clear evidence in the literature (in most experiments, both old and new) for such a development downstream of the imposition of an adverse pressure gradient. The exact nature of the evolution and the distance over which it occurs depends on the upstream boundary layer and the manner in which the APG is imposed. But far enough downstream the mean velocity profile in all cases becomes an inflectional point with the location of the inflection point corresponding quite closely to the observed peak in the streamwise turbulence intensity. This does not seem to have been previously noticed.

9:44AM MA.00009 Turbulence structure in non-zero pressure gradient boundary layers, JASON MONTY, ZAMBRI HARUN, IVAN MARUSIC, The University of Melbourne — We present an extensive database of single- and multi–probe hot-wire measurements of streamwise velocity acquired in zero, adverse and favourable pressure gradient turbulent boundary layers. The primary aim of this investigation was to characterise the effects of pressure gradient on the structure of turbulence at high Reynolds numbers. Specifically, we examine the changes to turbulence intensity, energy spectra and two-point correlations of streamwise velocity. By systematically varying the pressure gradient (PG) at a fixed Reynolds number (Re) we were able to isolate PG effects from Re effects. Results from the adverse pressure gradient case show a strong contribution to the energy spectra from length-scales of $\sim 3\delta$ ($\delta$ is the boundary layer thickness). This contribution is observed throughout the flow, even near the wall. Whether ‘superstructures’ (of length $\sim 6\delta$) exist or are modified in strong pressure gradients is unclear since the energy spectra indicate the dominance of $\sim 3\delta$-length structures in the logarithmic and outer regions. Two-point correlations indicate similar spanwise width-scales in the log region compared with the zero pressure gradient case, while the average structure becomes wider beyond the logarithmic region. Further results will be presented showing the effect of varying pressure gradient from favourable through zero to adverse at fixed Reynolds number.

9:57AM MA.00010 Analytical solutions of a quasilaminarized turbulent boundary layer, RIKI MINORU HOPKINS, RAUL BAYOAN CAL, Portland State University — Analytical solutions to the characteristic equation, arising from similarity analysis as proposed by Cal and Castillo (2008) describing a turbulent boundary layer subject to a strong favorable pressure gradient (FPG) approaching a quasilaminar state are found. By virtue of numerical analysis, solutions to this characteristic equation are obtained for several values of the pressure parameter, $\Lambda = -\frac{dU}{dx}\frac{d\delta}{dx}$, in addition to the Pohlhausen parameter, $K_s = \frac{\delta^3}{d\delta/dx}$. These solutions characterize the influence of the two parameters on a turbulent boundary layer subject to a strong FPG, and quantify these parameters for such flows with eventual quasilaminarization. Different cases are tested to observe the limits of these parameters. The analytical solutions obtained are compared to the experimental data obtained by Warnack and Fernholz (1998) A confirmation of the validity of this method and understanding of the influence of the remnants of the turbulence in the quasilaminar flow is assessed.


Tuesday, November 23, 2010 8:00AM - 10:10AM –
Session MB Turbulent Boundary Layers VIII Long Beach Convention Center 101B

8:00AM MB.00001 Status Update of the Flow Physics Facility at the University of New Hampshire. JIM FORSYTHE, JOE KLEWICKI, CHRIS WHITE, MARTIN WOSNIK, University of New Hampshire — The Flow Physics Facility at UNH is a unique high Reynolds number boundary layer wind tunnel. The facility uses a long length test section (72 m long, with a 2.5m x 6m cross section) to obtain a Reynolds number of about 50,000 based on boundary layer thickness and friction velocity. Since the tunnel uses a large development length and low speed (15-30 m/s) to create the boundary layer, the small scales of turbulence remain large enough to be measured with currently available instrumentation, enabling resolution of the entire turbulent spectrum at real-world scale Reynolds numbers. Phase I of the project has been constructed, enabling a test section speed of 15 m/s using an open-circuit design. Phase II has undergone preliminary design, with a funding request submitted, and would add a closed circuit and raise the maximum speed to 30 m/s. An adjustable ceiling allows is used to maintain a zero pressure gradient as the boundary layer grows down the length of the tunnel. A description of the facility’s attributes as well as preliminary measurements characterizing the test section flow will be presented.

1 Work funded by grants from NSF, ONR, and UNH
8:13AM MB.00002 Large-eddy simulation of the zero pressure gradient, turbulent boundary layer\textsuperscript{1}, MICHIO INOUE, D.I. PULLIN, Caltech — Large-eddy simulations (LES) of the zero-pressure gradient, smooth-wall, flat-plate turbulent boundary layer are presented. The LES combines the stretched-vortex, subgrid-scale (SGS) model with the tailored, near-wall model designed to incorporate anisotropic vorticity scales in the presence of the wall. Specifically, an approximate analytic integration of the stream-wise momentum equation across the near-wall layer, with inner-scaling used to reduce inertial terms, leads to a hyperbolic partial differential equation for the wall shear stress. This is coupled to an SGS model of streamwise, attached vortices in the presence of the wall, constructed to capture the principal dynamical behavior of longitudinal vortices in wall-normal transport of streamwise momentum. The result is an effective slip-velocity boundary condition for the LES at a raised “virtual wall” together with a dynamical calculation of the Kármán constant. Presently we demonstrate LES of the spatially developing, turbulent boundary layer at Reynolds numbers $Re_g$ based on the free-stream velocity and the momentum thickness in the range $Re_g = 10^3 - 10^5$. At large $Re_g$, the calculated skin-friction coefficient agrees well with the Coles-Fernholz relation.

\textsuperscript{1}Supported by the NSF.

8:26AM MB.00003 Combined Compact Difference Numerical Method for Simulation of Boundary Layer Turbulence Transition in the Non-Linear Stage, JIM CHEN, WEIJIA CHEN, Nanyang Technological University — The non-linear stage of boundary layer turbulence transition is investigated by solving the Vorticity Transport Equation using a 12th-order discretization of the spatial derivatives in uniform grids and a 4th-order 5-6 alternating stages Runge-Kutta method for temporal integration. The spatial and temporal schemes are optimized together for the downstream convective term to achieve better spectral resolution. In this method, the downstream wave number spectrum is divided into two parts: the first part preserves low dispersion and dissipation errors for accurate simulations of the physical waves; the second part generates strong numerical dissipation to suppress numerical grid-mesh oscillations. In addition, a multigrid method is used to accelerate the convergence of solving the velocity Poisson’s equation. Results of the simulations show that nonlinear wave interactions, generation, and amplification can be realized.

8:39AM MB.00004 Numerical simulation of negative Magnus force on a rotating sphere, MASAYA MUTO, MAKOTO TSUBOKURA, NOBUYUKI OSHIMA, Hokkaido University — Flow characteristics and fluid force on a sphere rotating along with axis perpendicular to mean air flow were investigated using Large Eddy Simulation at two different Reynolds numbers of 10,000 and 200,000. As a result of simulation, opposite flow characteristics around the sphere and displacement of the separation point were visualized depending on the Reynolds number even though the sphere rotates at the same rotation speed according to the Reynolds number. When Reynolds number is 10,000, flow characteristics agree with the flow field explained in the Magnus effect. However sphere rotates at the same rotation speed while increasing Reynolds number to 200,000, separation point moves in opposite direction and wake appears in the different direction. The reason of the negative Magnus force was discussed in terms of the boundary layer transition on the surface.

8:52AM MB.00005 Comparison of Intermittency Detection Algorithms in a Transitional Boundary Layer, JORDAN NULL, MARK MCQUILLING, Saint Louis University — Intermittency algorithms evaluate the maturity of the laminar-to-turbulent transition process by assessing how often a flow is turbulent. This work compares four different intermittency algorithms using thermal anemometry data sets. These data were acquired above the suction surface of a transitional low-pressure turbine airfoil at three chord Reynolds numbers of $2.5 \times 10^5$, $5.0 \times 10^5$, and $7.5 \times 10^5$. Comparisons between the Hedley-Keffer, Volino-Hultgren, Clark, and MTERA algorithms show that using one algorithm over another could lead to an improper interpretation of the transition physics occurring through the flowfield. Algorithms appear to signal the onset of transition in similar locations, although the surface-normal and surface-tangential extents and magnitudes of intermittency can vary considerably throughout the transitional region. Comparisons of the anemometry data to Coles’ Law of the Wake turbulent similarity velocity profiles also provide insight into the degree of correlation between algorithm intermittencies and accepted turbulent velocity profiles.

9:05AM MB.00006 Mean Dynamics of Channel Flow Transition, JOE KLEWICKI, University of New Hampshire, JOHN ELSNIT, University of Utah, DAN MAYNES, Brigham Young University, TIM AMEEL, University of Utah — The redistribution of mean momentum and vorticity are explored for laminar-to-turbulent transition in fully developed channel flow. A central aim is to better understand how the mean dynamical mechanisms representative of the fully turbulent regime are first established. Specifically, the mean dynamics of channel flow transition evolve from a simple balance of two differential forces at every point to a balance involving three terms that forms into a complex four layer structure. Primary considerations stem from the emergence of the effects of turbulent inertia as reflected by the appearance of the Reynolds stress gradient in the mean dynamical equation. The experimental and DNS results presented support a scenario in which the initial instabilities lead to a Reynolds stress distribution that is generally localized in space, and that subsequently spreads both inward and outward from the peak value. A characteristic feature of this distribution is the juxtaposition of positive and negative Reynolds stress gradients — a dynamical feature that persists for all higher Reynolds numbers. Connections are drawn between the mechanisms initiated during transition and those responsible for mean profile behaviors at much high Reynolds numbers.

9:18AM MB.00007 Mean momentum balance evolution in boundary layer transition, RACHEL EBNER, University of New Hampshire, XIAOHUA WU, Royal Military College of Canada, JOSEPH KLEWICKI, University of New Hampshire — The mean momentum balance in the high Reynolds number turbulent boundary layer has a four layer structure. This structure reflects a specific magnitude ordering of the underlying dynamical mechanisms. The observed properties of the mean velocity and Reynolds stress profiles follow directly from this ordering of terms. The recent DNS of Wu and Moin (JFM 630, 2009) is used to explore how the four-layer structure first forms. Specific and mathematically well-justified criteria are employed to identify the minimum Reynolds number at which the ordering of terms characteristic of the high Reynolds number state is first established. Physically, this is a consequence of the inward localization of the mean viscous force in concert with the outward localization of mean inertia. Comparisons indicate that, while the characteristic four layer structure for boundary layers and channels is very similar at high Reynolds numbers, the approach to these similar states occurs by a different route and at significantly different rates.

9:31AM MB.00008 Kelvin-Helmholtz-like instability of turbulent flows over riblets\textsuperscript{1}, RICARDO GARCÍA-MAYORAL, JAVIER JIMÉNEZ, U. Politecnica Madrid and CTR Stanford — The turbulent drag reduction due to riblets is a function of their size and, for different configurations, collapses well with a length scale $l_g = (A_g)^{1/2}$, based in the groove cross-section $A_g$. The initially linear drag reduction breaks down for $l_g \approx 11$, which agrees in our DNS with the previously reported appearance of quasi-two-dimensional spanwise rollers immediately above the riblets. They are similar to those found on porous surfaces and plant canopies, and can be traced to a Kelvin-Helmholtz-like instability associated with the relaxation of the impermeability condition for the wall-normal velocity. The extra Reynolds stress associated with them accounts quantitatively for the drag degradation. An inviscid model for the instability confirms its nature, agreeing well with the observed perturbation wavelengths and shapes. The onset of the instability is determined by a length scale $L_w$, that, for conventional riblet geometries, is proportional to $l_g$. The instability onset, $L_w \geq 4$, corresponds to the empirical breakdown point $l_g \approx 11$.

\textsuperscript{1}Funded by AVERT and CICYT.
9:44AM MB.00009 The Role of Turbulent Scales in a Rough and Smooth Surface Wind Turbine Blade¹. V. MALDONADO, S. TORRES-NIEVES, L. CASTILLO, Rensselaer Polytechnic Institute, C. MENEVEAU, Johns Hopkins University — In the present research, a 2-D (constant chord) wind turbine blade section based on an S809 airfoil was manufactured and tested at Johns Hopkins University in the closed return subsonic Corsin wind tunnel. The blade was covered with a 24-grit aluminum oxide abrasive sheet for the rough surface measurements. Smooth-wall blade measurements were also performed. Turbulence was generated using an active grid placed 5.5 m upstream of the blade. A free stream velocity of 10 m/s corresponding to a Reynolds number of 1.68x10^7 and angles of attack of 0 and 16 degrees (before and after flow separation) were selected, in order to study the effects of free stream turbulence on the aerodynamics and development of turbulent scales on the wind turbine. Global flow measurements such as mean velocity and Reynolds stresses were taken using Particle Image Velocimetry (PIV) and the pressure distribution around the suction and pressure surfaces of the 2-D blade with and without turbulence was acquired for angles of attack of 0 and 16 degrees. Initial results suggest that for the smooth surface blade at an angle of attack of 0 degrees, turbulence decreases lift production from a lift coefficient of 0.067 to 0.018, while at 16 degrees, turbulence enhances lift from 0.82 to 0.92.

¹ NSF-DGE-0742436 GK-12: Building Bridges from High School to Graduate School- Inspiring Students Through Discovery-Based Activities in Energy and the Environment.

9:57AM MB.00010 Subgrid Scale (SGS) Flow Structures and Energy Flux in a Rough-wall Channel Flow¹. JOSEPH KATZ, JIARONG HONG, CHARLES MENEVEAU, Johns Hopkins University, MICHAEL SCHULTZ, United States Naval Academy — This study examines interactions among turbulence structures of different scales based on high resolution PIV data obtained in a rough-wall channel flow, with δ/k ≈ 50 (k is roughness height), k^2 = 90-150 and Reynolds numbers of Re_τ ≈ 3520-5360. Top-hat spatial filtering with filter length scale of Δ = 1k, 3k, 6k divide the turbulence to roughness, intermediate and large scale motions, respectively. The SGS energy flux increases substantially with length scale and decreasing distance from the wall, especially in the roughness sublayer. The latter persist even when this flux is scaled with the local TKE production rate, which also peaks near the wall. Dissipation of energy is particularly high in the 1-3k range everywhere, especially in the roughness sublayer. Non-local transport, i.e. direct energy flux from large to roughness scales, which circumvents the typical cascading process, also increases rapidly near the wall. Conditional sampling indicates that this non-local flux is associated with inclined large scale shear layers (coherent structures) residing in the outer part of the boundary layer, which as our earlier data indicate, transport roughness scale turbulence to the outer layer.

¹ Sponsored by ONR

Tuesday, November 23, 2010 8:00AM - 9:31AM – Session MC Turbulence Simulations III Long Beach Convention Center 102A

8:00AM MC.00001 Simulation of Strong Shock and Turbulence Interactions using High-Order Shock-Fitting Algorithms¹. PRADEEP RAWAT, XIAOLIN ZHONG, University of California Los Angeles — We present results for Direct Numerical Simulations (DNS) of interactions of shock waves with realistic isotropic turbulence using shock-fitting schemes that are highly accurate and stable even for very strong shocks. We consider interaction of normal shocks of mean Mach numbers M_1 = 2 – 20 with incoming isotropic turbulence of turbulent Mach number, M_τ = 0.12 – 0.38, and Reynolds number based on Taylor microscale, Re_τ = 7 – 40. New trends are observed in turbulent statistics for the shocks stronger than those considered in previous studies. Amplifications in streamwise Reynolds stress values downstream of the shock are found to be initially decreasing as Mach number is increased but for stronger than Mach 8 shocks this trend reverses. We also observe that vorticity fluctuations return to isotropy behind the shock, but increasing Mach number of incoming flow delays this return to isotropy. Taylor microscales decrease as flow passes through a shock wave and amplification factors agree well with the linear theory results. Overall, the results generally confirm the findings by earlier studies but show new trends for stronger shocks than those considered in studies of the past.

¹ The research is supported by DOE-SciDAC grant DE-FC02-06ER25797:A002 and used computer resources of NERSC and TeraGrid.

8:13AM MC.00002 DNS of Shock / Isotropic Turbulence Interaction¹. NATHAN GRUBE, University of Maryland, ELLEN TAYLOR, PINO MARTIN, University of Maryland — We discuss DNS of Shock / Isotropic Turbulence Interactions (SITI). We vary the incoming turbulence Mach number up to 0.8 and the convective Mach number up to 5 in order to determine their effects on the interaction. These cases are challenging due to the presence of shocklets in the incoming turbulence as well as significant motion of the main shock. Shock-capturing must be used at all points while still maintaining low enough numerical dissipation to preserve the turbulent fluctuations. We use the linearly- and nonlinearly-optimized Weighted Essentially Non-Oscillatory (WENO) method[1,2]. Particular attention is paid to the inflow boundary condition, where we find the use of snapshots of “frozen” turbulence from decaying isotropic box simulations to be unsatisfactory. We instead use time-varying inflow data generated by a separate forced isotropic turbulence simulation with a specified convection speed. This allows us to access flow conditions where the assumptions of Taylor’s Hypothesis are not met. 1.) Martin, M.P., Taylor, E.M., Wu, M., and Weirs, V.G., JCP 220(1) 270-89, 2006. 2.) Taylor, E.M., Wu, M., and Martin, M.P., JCP 223(1) 384-97, 2007.

¹ Funded by AFSOR Grant Number AF 9550-09-1-0464 and AFRL.

8:26AM MC.00003 Large-eddy simulations of the shock-turbulence interaction canonical problem¹. IVAN BERMEJO-MORENO, JOHAN LARSSON, SANJIVA LELE, Center for Turbulence Research, Stanford University — We present results of large-eddy simulations of the interaction between a nominally planar shock wave and incoming isotropic turbulence passing through it. The numerical hybrid method in use combines fifth-order WENO and sixth-order central finite-difference schemes in a structured grid, with a sensor that switches between both schemes near shock waves. We compare results obtained with different SGS models, focusing on their performance in the relaxation region immediately downstream of the shock wave, where the effects of non-equilibrium and anisotropy are most noticeable. SGS models under evaluation include pure and mixed dynamic eddy-diffusivity models (with gradient and similarity mixed terms, and different types of eddy-diffusivities), as well as the structure-based stretched-vortex model. Extensions of existing models are proposed to improve their performance, evaluated through comparison of time-averaged turbulence quantities with filtered DNS results (see Phys. Fluids 21, 120101 (2009)).

¹Supported by the Department of Energy under the SciDAC II program.
et al. and an over-expanded supersonic jet injected into a subsonic crossflow, where the flow conditions are based on Santiago et al.'s (1997) and Beresh et al.'s (2005) experiments, respectively. A finite volume compressible Navier–Stokes solver developed by Park & Mahesh (2007) for unstructured grids is used. The simulations successfully reproduce experimentally observed shock systems and flow vertical structures such as the barrel shock, Mach disk, horseshoe vortices that wrap up in front of the jet and the counter rotating vortex pair (CVP) downstream of the jet. The dynamics of these flow structures are discussed, as well as the influence of grid resolution and the effect of inflow turbulence. The time averaged flow fields are compared to the experimental results, and reasonable agreement is observed.

This work has been supported by the NDSEG fellowship, PSAAP, and NASA.

8:52AM MC.00005 Simulations of High Speed Turbulent Jets in Crossflow, KRISHNAN MAHESH, University of Minnesota — Numerical simulations are used to study an under-expanded sonic jet injected into a supersonic crossflow and an over-expanded supersonic jet injected into a subsonic crossflow, where the flow conditions are based on Santiago et al.'s (1997) and Beresh et al.'s (2005) experiments, respectively. A finite volume compressible Navier–Stokes solver developed by Park & Mahesh (2007) for unstructured grids is used. The simulations successfully reproduce experimentally observed shock systems and flow vertical structures such as the barrel shock, Mach disk, horseshoe vortices that wrap up in front of the jet and the counter rotating vortex pair (CVP) downstream of the jet. The dynamics of these flow structures are discussed, as well as the influence of grid resolution and the effect of inflow turbulence. The time averaged flow fields are compared to the experimental results, and reasonable agreement is observed.

This work is supported by the National Science Foundation under grant CTS-0828162.

9:05AM MC.00006 The decay of forced rescaling modes in a Mach 3 turbulent boundary layer, YIN-CHIU KAN, IZZAAK BEEKMAN, STEPHAN PRIEBE, PINO MARTIN, University of Maryland — We introduce a new, Mach 3, compressible, turbulent boundary layer (TBL) spatial direct numerical simulation (SDNS), with a streamwise length of 506, increasing to 4000 at the outlet, with the boundary layer thickness, δ, nearly doubling from the inlet to the outlet. The inflow is computed using an auxiliary DNS with a rescaling length of 8δ. We examine the evolution of turbulence statistics as the boundary layer grows. In particular, we scrutinize the effects of rescaling and the non-stationarity of the flow. We wish to determine how far downstream the flow must travel to sufficiently “forget” the effects of rescaling. The effect of rescaling is of particular interest when investigating low frequency and large scale phenomena, such as coherent flow structures. These large coherent structures are on the order of 10δ in streamwise extent, and have been found at similar conditions to the present study. We set this data we set to address and quantify the role of rescaling and the rate at which the flow will forget artificial forcing.

1Supported by NASA Cooperative Agreement NNX08AD04A.

9:18AM MD.00007 DDES of shock wave/turbulent boundary layer interaction, PATRICIA CORONADO, MARCEL ILIE, University of Central Florida — The detached-eddy simulation (DES) model, which is a hybrid RANS and LES method, aims to solve the intensive CPU requirement of LES. Thus, near the solid surface within a wall boundary layer, the unsteady RANS model is realized, while away from the wall surface, the model automatically converts to LES. The delayed-detached eddy simulation (DDES) was proposed by Spalart in 2006 to improve the DES model previously developed. The transition from the RANS model to LES in DES is not grid spacing independent, therefore a blending function is introduced to the recently developed DDES model to make the transition from RANS to LES grid spacing independent. The present research concern the study of the shock/wave/turbulent boundary layer interaction using delayed-detached eddy simulation (DDES) model with a low diffusion E-CUSP (LDE) scheme with fifth-order WENO scheme. The first case studied using DDES is a 3D transonic channel with shock/turbulent boundary layer interaction. The second case studied consists of a 3D transonic inlet-diffuser. Both results are compared with experimental data. The computed results of the transonic channel agree well with experimental data. The results show that DDES simulation provides improved results for the shock wave/turbulent boundary layer interaction compared to those of its predecessor the detached-eddy simulation.

Tuesday, November 23, 2010 8:00AM - 9:44AM –
Session MD Turbulence Simulations IV Long Beach Convention Center 102B

8:00AM MD.00001 LES one-way coupling of nested grids using scale similarity model, KOJIRO NOZAWA, Shimizu Corp., TETSURO TAMURA, Tokyo Institute of Technology — The method for coupling between nested grids with turbulence energy smoothly transferred is proposed for LES turbulent flows. In this method fluctuating velocity simulated in a coarse grid is imposed to a fine grid. As a result, time-sequential data of the grid-scale velocity fluctuation of the fine grid can be obtained utilizing the scale similarity concept [J. Bardina, J. H. Ferziger and W. C. Reynolds, AIAA Paper, No.80-1357, (1980)]. The a-priori test of a turbulent boundary layer flow over a rough surface is conducted to validate this method. In order to fulfill simulations of spatially developing turbulent boundary layer flows we apply the quasi-periodic boundary condition to the streamwise direction [K. Nozawa and T. Tamura, Proc. of the Turbulent Shear Flow Phenomena, vol.2, 443-448, (2001)]. In the test coarsely resolved velocity data which is generated filtering finely resolved LES data are applied for directly reproducing grid-scale components of the coarsely resolved LES. The reproduced fluctuating velocity agrees well with the true value which can be obtained by subtracting the generated coarsely resolved velocity data from the finely resolved LES data. Also, the spectra of the reproduced streamwise fluctuation velocities at higher wave number range corresponding to the fine mesh size fit to the -5/3 power law for the inertial subrange. This method is expected to appropriately combine the meso-scale meteorological model with the LES model of urban scale.

8:13AM MD.00002 Formulation of smoothed-particle hydrodynamics method for turbulent free-surfaceflows, AKIHIKO NAKAYAMA, HIROSHI INOKUMA, Kobe University, KENTA IKENAGA, Toyota Technical Development — The Smoothed Particle Hydrodynamics (SPH) method is proving useful to compute various flows involving large deformation of flow field such as wave breaking and motion of solid bodies in fluids. The effects of turbulent fluctuations that are important in most large-scale flows in engineering and environmental applications have not been studied in the extent and the degree that conventional simulation methods like Large Eddy Simulation (LES) have been studied. We try to formulate the method as filtering in the moving frame of reference and identify what are exactly the effects of turbulent fluctuations that are resolved by this smoothed particle representation and show a method of modeling the effects. A few examples of numerical calculation results are presented to show the effectiveness of the proposed formulation.
8:26AM MD.00003 Turbulent modeling for low speed compressible flow, CHUNGANGLI, National Chiao Tung University, Taiwan, J.A. DOMARADZKI, University of Southern California, WUSHUNG FU, National Chiao Tung University, Taiwan — An investigation of turbulence models at high Reynolds numbers is conducted. The numerical code uses the Roe scheme and preconditioning matrix and dual time stepping are adopted for economizing the computational time and improving convergence properties. In order to validate the code, DNS of the turbulent channel flow are performed at Reynolds numbers, based on the friction velocity, of 180 and 500. The results for the mean velocity profiles and turbulent intensities are in good agreement with the benchmark DNS data obtained by spectral codes. The same code is used to perform LES with different models, among them the classical Smagorinsky model and the Truncated Navier Stokes (TNS) method, and comparisons are made with databases for high friction velocity Reynolds numbers of 1000 and 2000.

8:39AM MD.00004 Numerical computations of turbulent flows; LES/SAS comparison, MARCEILILIE, University of Central Florida, STEFAN LLEWELLYN SMITH, University of California, San Diego — In aerodynamics, the unsteady fluctuations of the flow field can have a significant influence on stalled flow characteristics, or on the forces acting on different parts of the aircraft. In non-aerodynamic flows, there is a multitude of mixing problems such as piston engines or turbine blade cooling where steady Reynolds-Averaged Navier-Stokes (RANS) solutions are not adequate. On the other hand the unsteady Reynolds-Averaged Navier-Stokes (URANS) has proven to be insufficient. This is due to the highly dissipative nature of standard URANS. The use of Large Eddy Simulation (LES) methods is often not practical, due to the requirement of very fine grid resolution near walls. Direct Numerical Simulations (DNS) compute the flow field without further simplifications. However, due to a wide range of length and time scales present in turbulent flows, the use of DNS is still limited to low-Reynolds-number flows and relatively simple geometries. To combine the advantages of a URANS with the higher resolution of a LES, hybrid methods such as Detached Eddy Simulation (DES) or Scale Adaptive Simulation (SAS) are preferred. The present research concerns the suitability of SAS for the computation of highly separated flows. The results show that SAS is a promising approach for the computation of massively separated flows.

8:52AM MD.00005 Wall-modeling for large-eddy simulation of high Reynolds number supersonic flows1, SOSHI KAWAI, JOHAN LARSSON, SANJIVA LELE, Center for Turbulence Research, Stanford University — We present an idea of approximate wall-boundary-condition approach with dynamic procedure for large-eddy simulation of Mach 3 supersonic turbulent boundary layer at various Reynolds numbers $Re_x = 2 \times 10^4$, $10^5$ and $10^6$ on a flat plate. This wall-model is the extension of previous work by Wang and Moin [Phys. Fluids, 14, 2043 (2002)] for incompressible flows to compressible flows. We note that the present study is both the first extension of the dynamic concept to compressible flows and also the first test at high Reynolds number flows. The present study also revisits the issue of numerical errors near wall-region on outer-layer coarse LES mesh. The numerical results are compared with wall-resolved LES data (at low Reynolds number case) and available experimental data (at high Reynolds number case).

1This work is supported by NASA Fundamental Aeronautics Program - Hypersonics Project (Grant NNX08AB30A).

9:05AM MD.00006 A buoyancy-adjusted extension of the stretched-vortex subgrid-scale model, DANIEL CHUNG, GEORGIOS MATHEOU, Jet Propulsion Laboratory/California Institute of Technology — We present a buoyancy-adjusted extension of the stretched-vortex subgrid-scale (SGS) model suitable for large-eddy simulation (LES) of stratified flows. The model remains free of parameters and is consistent with features of anisotropic mixing frequently observed in stratified flows. The vortex-based construction naturally constrains the mixing in the horizontal provided the vortex alignment is favorable even at high gradient Richardson numbers. We will compare the LES results with direct numerical simulation (DNS) of homogeneous stably stratified flows.

9:18AM MD.00007 Scaling laws in helical rotating turbulence: do they change with Reynolds number? , A. POUQUET, NCAR, J. BAERENZUNG, INPG, P. MININNI, UBA, D. ROSENBERG, NCAR — In rotating turbulence, the presence of helicity leads to significant differences when compared to flows without global velocity-vorticity correlations, as seen using direct numerical simulations (DNS) with up to 1536$^3$ points, down to Rossby numbers $Ro \approx 0.06$. Long-lived laminar structures and turbulent vortices co-exist. The energy undergoes both an inverse and a direct cascade, the latter being self-similar with spectrum $E(k) \sim k^{-5/3}$ and transfer rate $\epsilon$ and dominated by the helicity cascade (spectrum $H(k) \sim k^{-h}$, transfer $\hat{\epsilon}$). This points to the existence of a new small parameter, $\epsilon/|La|$, with $La$ a characteristic length. We also find that $\epsilon + h = 4$, taking into account the inertial wave mediation of nonlinear helicity transfer to small scales, with $e \neq h$ at the intermediate Reynolds numbers at which we compute. Using an isotropic model based on the Eddy Damped Quasi-Normal Markovian closure, and including the contribution of helicity to eddy viscosity and eddy noise, we show that we can recover the DNS results at substantially lower costs and that, at fixed Reynolds number, strong rotation leads to the $e + h = 4$ regime whereas $e = h = 5/3$ when increasing the Reynolds number at fixed rotation rate.

9:31AM MD.00008 Analysis of Structure Functions for the Turbulent Ekman Layer Direct Simulation, SCOTT WAGGY, SEDAT BIRINGEN — A direct numerical simulation of the low-Reynolds number turbulent Ekman layer was performed to assess the validity of Kolmogorov similarity laws in rotating turbulent flows. The three dimensional mean flow exhibited by the Ekman layer offers complex energy transfers not encountered in simple two-dimensional turbulent flows with one main mean shear direction. Time averaged 2nd order velocity structure functions were calculated to determine the extent of the inertial subrange at low Reynolds numbers. In addition, the constant $C_{9/7}$, a universal constant of the structure functions, was compared with non-rotating boundary layers to analyze their applicability to different flows. The degree to which higher order structure functions abide by Kolmogorov’s scaling was also analyzed for 3rd and 4th order structures.

Tuesday, November 23, 2010 8:00AM - 10:10AM – Session ME General Fluid Dynamics III Long Beach Convention Center 102C

8:00AM ME.00001 Upstream Drafting of a Flexible Body by its Downstream Neighbor, TEIS SCHNIPPER, Applied Mathematics Laboratory, Courant Inst., New York University and Dept. of Physics and Center for Fluid Dynamics, Technical University of Denmark, JUN ZHANG, Applied Mathematics Laboratory, Courant Inst., and Dept. of Physics, New York University — It is common knowledge that an upstream body influences its downstream neighbors in an open flow. This is often referred to as flow drafting or slipstreaming (either in the air or in water). In this talk, we present an experimental study on how the motion of a flapping flag is strongly affected by a downstream neighbor. In a flowing soap film tunnel we introduce, in turn, passive as well as kinematically driven bodies in the wake of an otherwise freely flapping flag. We show how the flapping frequency and drag on the leading flag can be significantly manipulated by the downstream neighbor.
8:26AM ME.00003 Adaptive Control of the Generalized Korteweg-de Vries Burgers Equation
NEJIB SMAOUI, ALAA EL-KADRI, MOHAMED ZRIBI, Kuwait University — The adaptive boundary control problem of the generalized Korteweg-de Vries-Burgers (GKdVB) equation when the spatial domain is [0,1] is considered. Three adaptive control laws are designed for the GKdVB equation when either the kinematic viscosity \( \nu \) or the dynamic viscosity \( \mu \) is unknown, or when both viscosities \( \nu \) and \( \mu \) are unknowns. Using the Lyapunov theory, the L2-global exponential stability of the solutions of this equation is shown for each of the proposed control laws. Also, numerical simulations based on the Finite Element method (FEM) are given to illustrate the analytical results.

8:39AM ME.00004 Effect of orifice eccentricity on the Vortex size downstream orifice cascade in laminar duct flow
AHMED ABOU EL-AZM ALY, MOHAMED METWALLY, HOSSAM ABDEL KADER, Military Technical College, Cairo, Egypt — Vortex formation and shedding downstream obstructions may be assumed to be one of the main sources of induced vibration and noise in pipes. The vortex size may be estimated from its reattachment length downstream these obstructions. Here, vortex formation downstream orifice cascade in a laminar duct flow has been investigated numerically. The vortex reattachment length downstream the second orifice and the circulation have been studied for different geometry parameters: orifice eccentricity, orifice to duct height ratio and orifice cascade interdistance. The results showed an optimum value (minimum vortex size) is a function of duct geometry and orifice to duct height, a correlation function has been deduced to illustrate this relation. The results showed that the control of these parameters may be of special interest, to reduce the generated vortex size.

9:05AM ME.00006 Physical Gelation of a Nano-Composite Soft Glass
H. HENNING WINTER, National Science Foundation and University of Massachusetts Amherst, KATIE T. LANIA, PEI LI, University of Massachusetts Amherst, XIAOLIANG WANG, Nanjing University, China — Materials in Nature often gain their functionality from being composite on the smallest scale. This is mimicked in manmade nano-composites which profit from the large specific surface area of thin solid enclosures (examples: clay leaves or graphene). Here we use rheology to examine the slow ripening of an out-of-equilibrium model system (“soft glass”) that consists of clay particles that swell, break up, and eventually exfoliate into randomly oriented clay leaves through the action of end-functionalized (“sticky”) polymer molecules. The nano-composite serves as model soft glass in search of regular patterns in the non-equilibrium dynamics in the approach of equilibrium. Experiments on the model system suggest a scaling relation for the time-resolved viscoelasticity of physical gelation (Macromolecules 43:1901, 2010). Experiments on a wider group of soft glasses is in progress with the objective of confirming or rejecting universality of the well findings. The experimental protocol includes time-resolved rheometry (Rheol Acta 33:385-397, 1994) and rescaling of data.

9:57AM ME.00010 Modifying intake flow to increase EGR tolerance in an Internal Combustion Engine
1 DANIEL RUBIO, University of Central Florida, MEBOUGNA DRABO, PAUL PUZINÄUSKAS, University of Alabama — The worldwide effort to reduce vehicle emissions and increase fuel efficiencies has continuously intensified as the need to improve air quality and reduce fuel consumption becomes more acute. Exhaust gas recirculation (EGR) is a method that has long been employed to reduce combustion temperatures and therefore reduce thermal NOx formation and accommodate higher compression ratios and more optimum combustion phasing for improved efficiency. Generally the effective EGR level as a percent of trapped charge is limited by its affect on combustion stability. Inducing flow structures such as swirl, squish and tumble in the trapped charge have proven to extend this EGR limit in homogeneous charge spark-ignited engines at part load, but this enhancement has not been significantly studied at full loads in such engines. This research explored modifying the intake flow into an engine to create tumble and evaluate its effect at high loads in such engines. This exploration included characterizing the flow on a steady flow bench and quantifying the results using engine dynamometer tests.

1Work performed under REU site sponsored by NSF grant EEC 0754117.
8:00AM MF.00001 Hibernating turbulence, edge states and the Virk asymptote in channel flow of Newtonian and polymeric fluids, MICHAEL GRAHAM, Univ. of Wisconsin-Madison, LI XI, MIT — Turbulent channel flow of Newtonian and drag-reducing polymer solutions is studied computationally. Simulations in the minimal channel geometry reveal that, even in the Newtonian limit, there are intervals of “hibernating” turbulence that display many features of the universal maximum drag reduction (MDR) asymptote observed in polymer solutions: weak streamwise vortices, nearly nonexistent streamwise variations and a mean velocity gradient that quantitatively matches experiments (i.e. the Virk log-law). As viscoelasticity increases, the frequency of these intervals also increases, while the intervals themselves are unchanged, leading to flows that increasingly resemble MDR. Simulations in larger channel flow domains as well as turbulent boundary layers reveal spatiotemporally localized regions of active and hibernating turbulence, with hibernating turbulence becoming dominant as the level of viscoelasticity (and drag reduction) increases. Simulations of “edge states”, dynamical trajectories that lie on the basin boundary between turbulent and laminar flow, display characteristics that are similar to those of hibernating turbulence and thus to the Virk asymptote, again even in the Newtonian limit.

8:13AM MF.00002 Turbulent Drag Reduction by Polymers: A Theoretical Study on the Effect of Polymer Concentration, CHUNG YIN LEUNG, EMILY S.C. CHING, The Chinese University of Hong Kong — A recent theory on turbulent drag reduction by polymers in wall-bounded flows, which is based on the balance of momentum and energy, has been developed [I. Procaccia, V.L. L’ov, and R. Benzi, Rev. Mod. Phys. 80, 225 (2008)]. In this theory, the effect of the polymers is understood as a position-dependent effective viscosity. Using this theory, we have carried out a study on the effect of polymer concentration for both flexible and rigid polymers. We have calculated the profiles of the mean velocity and the Reynolds stress and investigated how the profiles change with the polymer concentration. We find some interesting relation between the maximum Reynolds stress and the position of the maximum for a large range of concentration. We have also calculated how the friction factor and the percentage of drag reduction vary with concentration. In this talk, we shall discuss our results and also compare them with experimental observations.

8:26AM MF.00003 On the correspondence between polymer-modified turbulence states and transitional states in Newtonian flows¹, YVES DUBIEF, School of Engineering, University of Vermont, Burlington, VT, CHRISTOPHER WHITE, Dept. of Mech. Eng., University of New Hampshire, Durham, NH — Polymer addition is known to produce drag in wall-bounded flows up to an asymptotic state called maximum drag reduction (MDR). The definition of MDR is still largely empirical and its uniqueness is a matter of debate. Using direct numerical simulations, a correspondence is first established between MDR and a specific state of transition in boundary layer flow. A model is derived as a function of the flow topology of the transitional Newtonian flow and the FENE-P model. The model is then extended to natural convection where heat transfer reduction (HTR) and augmentation (HTA) are observed as a function of polymer length. Yet, HTR and HTA are topologically equivalent and again correspond to a transitional state of Rayleigh Benard convection flow. This suggests that polymer-modified turbulence may be predictable as a function of the polymer solution’s properties and transitional states of the corresponding Newtonian flow.

8:39AM MF.00004 Parallel Large-Scale Computation of an Oldroyd-B Fluid Past a Confined Circular Cylinder in a Rectangular Channel using an Unstructured Finite Volume Method, MEHMET SAHIN, Istanbul Technical University — A new stable unstructured finite volume method is presented for parallel large-scale simulation of viscoelastic fluid flows. The numerical method based on side-centered finite volume method where the velocity vector components are defined at the mid-point of each cell face, while the pressure term and the extra stress tensor are defined at element centroids. The present arrangement of the primitive variables leads to a stable numerical scheme and it does not require any ad-hoc modifications in order to enhance the pressure-velocity-stress coupling. The log-conformation representation has been implemented in order improve the limiting Weissenberg numbers in the proposed finite volume method. The time stepping algorithm used decouples the calculation of the extra stresses from the evaluation of the velocity and pressure fields by solving a generalised Stokes problem. The present numerical method is verified for the three-dimensional flow of an Oldroyd-B fluid past a confined sphere in a cylindrical tube. Then the method is applied to the three-dimensional flow of an Oldroyd-B fluid past a confined circular cylinder in a rectangular channel. The computed results at relatively high Weissenberg numbers are discussed and compared to those obtained for Newtonian fluids.

8:52AM MF.00005 A viscoelastic drop falling through a viscous fluid, MUKHERJEE SWARNAJAY, KAUSIK SARKAR, University of Delaware — A viscoelastic drop falling through a Newtonian medium is simulated using a front tracking finite difference method. The drop viscoelasticity deforms the drop into an oblate shape. Further increase in viscoelasticity forms a dimple at the rear end of the drop. The dimple is a result of viscoelastic stresses which pull the drop interface towards the center. The dimple becomes increasingly prominent as Deborah number or the capillary number is increased. An approximate analysis is executed to model the stress development along the axis of symmetry, specifically its increase near the rear end that governs dimple formation. The model also suggests a shift of the maximum of the viscoelastic stresses toward the centre of the drop with increasing Deborah number. For even higher values of Deborah number, the interface cannot balance the viscoelastic stresses and the dimple grows to make the drop unstable. Unstable cases accompany a decrease in velocity because of the formation of a globular shape at the end of the dimple. This results in a sudden increase in the cross-sectional area of the drop and simultaneous decrease in the settling velocity. Finally, we determine the critical Deborah number for transition from stable to unstable cases for varying capillary number.

9:05AM MF.00006 Simulations of high Reynolds number wake transition in the presence of viscoelasticity, DAVID RICHTER, GIANLUCA IACCARINO, ERIC SHAQFEH, Stanford University — Using our three dimensional, time dependent finite volume code developed to compute non-Newtonian flows over a large range of Reynolds number (Re), we performed simulations of viscoelastic flow past a circular cylinder. Our focus was on elucidating elastic effects on transition to turbulence in the presence of viscoelasticity. The FENE-P constitutive model was used to describe the presence of polymers, and the numerical method employed was such that a large-range of rheological parameters (polymer length L, dimensionless Weissenberg number (Wl), and polymer concentration β) could be probed. We present a study of the viscoelastic effects on the inertial wake at high Re. Simulations were performed at Reynolds numbers of both 300 and 3900, and in each case we witness significant viscoelastic stabilization of structures typically seen in Newtonian flows. At Re = 300, the characteristic Newtonian mode A and mode B instabilities can either be weakened or completely suppressed based on the polymer extensibility L — an effect which has been further confirmed with linear stability analysis. Furthermore, at Re = 3900, even a small concentration of low extensibility polymers can have the ability to stabilize the shear layer (which has transitioned for pure Newtonian flow), and revert the wake structure back to one resembling the mode B instability, a state seen in Newtonian flows at much lower Reynolds numbers.
9:18AM MF.00007 Evolution of vortical structures in Newtonian and viscoelastic turbulent flows1, KYOUNGYOU N KIM, Hanbat National University, Daejon, South Korea; RADHAKRISHNA SURESHKUMAR, Syracuse University, Syracus, NY — To study the influence of dynamical interactions between turbulent vortical structures and polymer stress on turbulent friction drag reduction, a series of simulations were performed for channel flow at Reₘ = 395. The initial eddy extracted by the conditional averages for the Q2 event from fully turbulent Newtonian flow is self-consistently evolved in the presence of polymer stresses by utilizing the FENE-P model (finite extensible nonlinear elastic-Peterlin). The initial polymer conformation field is given by the solutions of FENE-P model equations for the Newtonian mean shear. For a relatively low Weissenberg number, defined as the ratio of fluid relaxation time to the time scale of viscous diffusion, (Wₑₑ = 50) the generation of new vortices is inhibited by polymer-induced counter torques, which results in fewer vortices in the buffer layer. However, the head of primary hairpin unaffected by the polymer stress. For larger values of Wₑₑ (≥100), the hairpin head becomes weaker and vortex auto-generation and Reynolds stress growth are almost entirely suppressed.

1KK would like to acknowledge the support from KISTI Supercomputing Center (No. KSC-2009-502-0013).

9:31AM MF.00008 Peristaltic pumping of solid particles immersed in a viscoelastic fluid1, JOHN CHRISPEL, LISA FAUCI, Tulane University — Peristaltic pumping of fluid is a fundamental method of transport in many biological processes. In some instances, particles of appreciable size are transported along with the fluid, such as ovum transport in the oviduct or kidney stones in the ureter. In some of these biological settings, the fluid may be viscoelastic. In such a case, a nonlinear constitutive equation to describe the evolution of the viscoelastic contribution to the stress tensor must be included in the governing equations. Here we use an immersed boundary framework to study peristaltic transport of a macroscopic solid particle in a viscoelastic fluid governed by a Navier-Stokes/Oldroyd-B model. Numerical simulations of peristaltic pumping as a function of Weissenberg number are presented. We examine the spatial and temporal evolution of the polymer stress field, and also find that the viscoelasticity of the fluid does hamper the overall transport of the particle in the direction of the wave.

9:44AM MF.00009 Examining the coil-stretch transition in flexible polymers, PATRICK UNDERHILL, RANGARAJAN RADHAKRISHNAN, Rensselaer Polytechnic Institute — The behavior of polymer solutions in elongational flow is important in many applications. An especially important property is the strain rate hardening resulting from the coil-stretch transition. Predictions of the coil-stretch transition and hysteresis have been verified by visualizing single molecules of double-stranded DNA (ds-DNA). The same behavior has not yet been directly observed in single molecule studies of synthetic polymers or more flexible biopolymers such as single-stranded DNA. Current theories of flexible polymers predict these other polymers will behave in a similar way to ds-DNA. However, we have very recently predicted that these other polymers could have dramatically different behavior; the coil-stretch transition can be eliminated under some conditions. For this purpose, we have altered the common bead-spring chain polymer models and simulated their response in flow using Brownian dynamics (BD). This new model we developed allows us to capture the importance of flexibility, entropic elasticity, hydrodynamic interactions, and solvent quality in an accurate and efficient way. This would have not been possible using conventional methods of including excluded volume as a repulsive interaction potential between beads; such a model would require such a large number of beads interacting that it would not be computationally tractable.

9:57AM MF.00010 The dynamics of a simple model for a yield stress fluid1, YURIKO RENARDY, Virginia Tech, KARA MAKI, University of Minnesota — A simple model for a yield stress fluid is obtained from Larson’s partially extending convection (PEC) strand model by replacing the zero shear stress limit for large shear rates with a non-negative limit (PECR), and augmenting with a Newtonian solvent (PECR-N). The constitutive behavior of PECR-N exhibits the typical non-monotonic shear stress versus shear rate behavior which allows for yielding to occur above a critical value of applied stress. The experimental determination of yield stress can be complicated by extremely slow yielding which may occur for a range of applied stresses. We therefore focus on the case where the elastic time scale is large compared with the viscous time scale and study the evolution of the conformation tensor for parallel shear flow with prescribed shear stress. The resulting dynamical system is solved both numerically and with asymptotic methods to clarify the different types of solution behavior. We find that multiple time scales are responsible for the path to transition from a fast curve, landing on a slow manifold, and escaping to yielded states which are steady or time-periodic. Novel solution types will be discussed.

1NSF-DMS, IMA (U. Minn.)

Tuesday, November 23, 2010 8:00AM - 10:10AM —
Session MG GFD: Atmospheric Flows III Long Beach Convention Center 103B

8:00AM MG.00001 A new anemometer for 2D atmospheric flow measurements in rough environments1, HENDRIK HEISSEL, MICHAEL HOELLING, JOACHIM PEINKE, ForWind - University of Oldenburg — One major downside of cup anemometry is the different response time for increasing and decreasing wind speeds, causing a systematic over-estimation of the mean wind speed under turbulent conditions. Especially under harsh environmental conditions like in offshore operation, the measuring principle leads to a wear of bearings causing a de-calibration over time and the requirement of regular maintenance. Therefore, we propose the newly developed sphere anemometer as a simple and robust alternative without any moving parts. The sphere anemometer consists of a flexible tube with a sphere mounted on top of it. The drag force acting on the sphere and its support causes a deflection, which is measured by means of a light pointer. Via calibration, this allows for simultaneous determination of wind speed and direction using only one sensor. In our contribution, we introduce the anemometer’s setup and it’s optimization towards offshore application. Additionally, experimental results obtained from wind tunnel measurements of turbulent flows are presented. Measurements under real wind conditions are compared to those of state-of-the-art wind speed sensors, such as cup and ultrasonic anemometers.

8:13AM MG.00002 Interaction of pollution plumes and discontinuous fields in atmospheric chemistry models, MAURICIO SANTILLANA, MICHAEL P. BRENNER, Harvard University; YEVGENIY RASTIGEYEV, North Carolina Agricultural and Technical State University, DANIEL J. JACOB, Harvard University — Atmospheric pollutants originate from concentrated sources such as cities, power plants, and biomass fires. They are injected in the troposphere where eddies and convective motions of various scales act to shear and dilute the pollution plumes as they are advected downward. Despite this shear and dilution, observations from aircraft, sondes, and satellites show that pollution plumes in the remote free troposphere can preserve their identity as well-defined layers for a week or more as they are transported on intercontinental scales. This structure cannot be reproduced in the standard Eulerian chemical transport models used for global modeling of tropospheric composition, instead, the plumes dissipate far too quickly. In this work, we study how the structure of plumes is modified when they cross discontinuities arising for example: from the moving day-night boundaries or from abrupt unresolved horizontal temperature changes (for example in horizontal ocean-land or ocean-ice transitions). Chemical reactions within the plumes depend strongly on photon availability and temperature, and thus, discontinuities in these variables lead to discontinuous changes in reaction rate constants.
9:05AM MG.00006 Laboratory simulations of cumulus cloud flows explain the entrainment anomaly, ROODAM NARASIMHA, SOURABH S. DIWAN, JNCSAR, Bangalore, DUVVURI SUBRAHMANYAM, CALTECH, K.R. SREENIVAS, JNCSAR, Bangalore, G.S. BHAT, IISC, Bangalore — In the present laboratory experiments, cumulus cloud flows are simulated by starting plumes and jets subjected to off-source heat addition in amounts that are dynamically similar to latent heat release due to condensation in real clouds. The setup permits incorporation of features like atmospheric inversion layers and the active control of off-source heat addition. Herein we report, for the first time, simulation of five different cumulus cloud types (and many shapes), including three genera and three species (WMO Atlas 1987), which show striking resemblance to real clouds. It is known that the rate of entrainment in cumulus cloud flows is much less than that in classical plumes - the main reason for the failure of early entrainment models. Some of the previous studies on steady-state jets and plumes (done in a similar setup) have attributed this anomaly to the disruption of the large-scale turbulent structures upon the addition of off-source heat. We present estimates of entrainment coefficients from these measurements which show a qualitatively consistent variation with height. We propose that this explains the observed entrainment anomaly in cumulus clouds; further experiments are planned to address this question in the context of starting jets and plumes.

8:39AM MG.00004 Statistical scale invariance in satellite observations of water vapor mixing ratio from the Atmospheric Infrared Sounder, KYLE PRESEL, The University of California, Berkeley, WILLIAM COLLINS, Lawrence Berkeley National Laboratory — Statistical scale invariance appears almost ubiquitously in fluid dynamical systems and often characterizes universal aspects of particular classes of flows. Perhaps the most famous instances of statistical scale-invariance in the atmospheric sciences are the Kolmogorov's -5/3 and Charney's -3 variance spectra for passive scalars and velocity in 3D and quasi-geostrophic turbulence respectively. Parameterizations of radiative transfer and clouds in global climate models (GCMs) depend on proper characterization of the spatial statistics of water vapor, which is not a passive scalar. Empirical investigations of the scale dependence of water vapor statistics have largely depended on aircraft observations, which are limited in spatial and temporal extent. We will present results from a structure function analysis of statistical scale invariance of water vapor mixing ratio fields as observed by the Atmospheric Infrared Sounder (AIRS) onboard NASA’s Aqua satellite and discuss the application of these results to the GCM parameterization problem.

8:26AM MG.00003 Punctuated changes in plant pathogen populations associated with passage of atmospheric Lagrangian coherent structures, SHANE ROSS, PHANINDRA TALLAPRAGADA, DAVID SCHMALE, Virginia Tech — The atmospheric transport of airborne microorganisms (e.g., plant pathogens) is poorly understood, yet necessary to assess their ecological roles in agricultural ecosystems and to evaluate risks posed by invasive species. The atmospheric transport of plant pathogens can be roughly divided into three phases: liberation of pathogen spores, drift (transport in the atmosphere) and deposition. If liberated spores escape into the planetary boundary layer, they could be transported over thousands of kilometers before being deposited. The drift phase is poorly understood, due to the complex nature of atmospheric transport and relative lack of observational data. In this talk, we present a framework of Lagrangian coherent structures to determine the important atmospheric transport barriers (ATBs) that partition the atmosphere and systematically organize the mesoscale transport problem. Using autonomous unmanned aerial vehicles, we measure the concentration of spores of a plant pathogenic fungus (Fusarium) sampled in the atmosphere above Virginia Tech’s Kentland Farm. We report correlations between concentrations of Fusarium with the local movement of ATBs determined from archived meteorological data.

8:52AM MG.00005 Analysis and numerical simulation of a laboratory analog of radiatively induced cloud-top entrainment, ALAN KERSTEIN, Sandia National Laboratories, HEIKO SCHMIDT, Technical University of Cottbus, RENAUD NEDELEC, Ecole Centrale Marseille, SCOTT WUNSCH, Johns Hopkins University Applied Physics Laboratory, BEN SAYLER, Black Hills State University — Numerical simulations using the One-Dimensional-Turbulence model are compared to water-tank measurements emulating convection and entrainment in stratiform clouds driven by cloud-top cooling. Measured dependences of the entrainment rate on Richardson number, molecular transport coefficients, and other experimental parameters are reproduced. Additional parameter variations suggest more complicated dependences of the entrainment rate than previously anticipated. A simple algebraic model indicates the ways in which laboratory and cloud entrainment behaviors might be similar and different.

9:18AM MG.00007 The evaporatively driven cloud-top mixing layer, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — Turbulent mixing caused by the local evaporative cooling at the top cloud-boundary of stratocumuli will be discussed. This research is motivated by the lack of a complete understanding of several phenomena in that important region, which translates into an unacceptable variability of order one in current models, including those employed in climate research. The cloud-top mixing layer is a simplified surrogate to investigate, locally, particular aspects of the fluid dynamics at the boundary between the stratocumulus clouds and the upper cloud-free air. In this work, direct numerical simulations have been used to study latent heat effects. The problem is the following: When the cloud mixes with the upper cloud-free layer, relatively warm and dry, evaporation tends to cool the mixture and, if strong enough, the buoyancy reversal instability develops. This instability leads to a turbulent convection layer growing next to the upper boundary of the cloud, which is, in several aspects, similar to free convection below a cold horizontal surface. In particular, results show an approximately self-preserving behavior that is characterized by the molecular buoyancy flux at the inversion base, fact that helps to explain the difficulties found when doing large-eddy simulations of this problem using classical subgrid closures.

8:31AM MG.00008 A study of the interactions between turbulence and small inertial droplets, COLIN BATESON, ALBERTO MOLINA, University of Washington, BOGDAN ROSA, LIAN-PING WANG, University of Delaware, ALBERTO ALISEDA, University of Washington — Understanding the dynamics of particles in turbulent flows is important to many engineering and environmental problems including spray atomization and cloud droplet growth and precipitation. Specifically, we have studied the effect of turbulence on droplet collision-coalescence in an effort to clarify its role in the process of warm rain formation. The cloud-top mixing layer is a simplified surrogate to investigate, locally, particular aspects of the fluid dynamics at the boundary between the stratocumulus clouds and the upper cloud-free air. In this work, direct numerical simulations have been used to study the evolution of water droplets in homogeneous, isotropic, slowly decaying grid turbulence. Droplets between 1 and 120 µm were injected into the wind tunnel and their diameter, position and velocity were measured at different distances downstream by Phase Doppler Particle Analysis (PDPA). Statistics of the radial distribution function (RDF), relative velocity distribution and settling velocity have been produced and analyzed. They will be compared to the same statistics computed from 3D hybrid direct numerical simulations (DNS) at similar Re. High-speed visualizations of the droplet dynamics will be explored in an effort to understand and quantify coalescence efficiency.

1Supported in part by the NSF under Grant No. DEB-0919088.

9:44AM MG.00009 Enhancement of coalescence due to droplet inertia in turbulent clouds.
STEVEN KRUEGER, University of Utah, ALAN KERSTEIN, Sandia National Laboratories — In the Explicit Mixing Parcel Model of mixing effects on cloud-
droplet evolution, turbulent advection of fluid is implemented by permutations ("triplet maps") of the fluid cells in chosen segments of the 1D domain, each
representing an individual eddy. This captures motions as small as the smallest turbulent eddies (Kolmogorov microscale), but there is important droplet-inertia
phenomenology, such as droplet clustering that increases droplet collision rates, at much smaller scales. We have developed and demonstrated a 3D triplet map
for droplets (and an associated drag-law representation) that captures clustering behaviors at small Stokes numbers St (such as those of cloud droplets). There
is excellent agreement between our results (for radial distribution functions and collision kernels) at small St and direct-numerical-simulation (DNS) results that
omit gravity, and good agreement with DNS results that include gravity. We are currently testing an extension of our model that is intended to broaden its
applicability to higher St, and we are using a collision-detection algorithm to simulate coalescence.

9:57AM MG.00010 Experimental study of droplet condensational growth in a wet/dry turbu-
ulent mixing layer, RYAN KEEDY, ALBERTO ALISEDA, University of Washington — Droplet condensational growth has been experimentally studied
as a function of supersaturation and turbulence. The experimental setup consists of two coaxial round jets, with a moist, warm inner jet, surrounded by an
anunnular sheath of cold air. Supersaturation is controlled by the relative humidity and temperature of the two streams, while the turbulence levels are determined
by the shear at the interface between the two jets. Turbulent mixing of the supersaturation field, as well as particle clustering in regions of low vorticity, are
expected to lead to large inhomogeneities in the growth rate and resulting discrepancies in the particle size distribution. Analysis of large data sets of droplet
behavior at different locations along the mixing layer and under different conditions of water vapor concentration, temperature and mixing intensity are used
to understand the dynamics of the interaction between the turbulent eddies and condensational droplet growth. A Phase Doppler Particle Analyzer (PDPA)
is used to collect statistics of droplet growth, and to characterize the turbulent velocity field. The goal of this study is to improve the understanding of the
competition between turbulent mixing replenishing the water content around the droplet and depletion of the vapor surrounding the droplet by condensation,
and to develop quantitative models that can be applied to the problem of edge mixing in clouds.

Tuesday, November 23, 2010 8:00AM - 10:10AM —
Session MH GFD: General II Long Beach Convention Center 103C

8:00AM MG.00001 Non-cohesive, unimodal sediment transport in non-hydrostatic dam-break
flow1, PATRICIO BOHORQUEZ, Universidad de Jaen — The mixture equations for non-cohesive, unimodal sediment transport in turbulent free-surface flow
are derived from the conditionally averaged Navier-Stokes equations. The mathematical similarity between the sediment volumetric concentration $\beta$ and the
water phase indicator function $\gamma$ is highlighted in the present model. We take advantage of this fact to formulate an explicit Finite Volume Method in which the
pressure equation is formulated as the Schur complement in a segregated pressure-based solver. The numerical scheme was implemented into OpenFOAM®,
an open source software tailored for Computational Continuum Mechanics. The capabilities to account for non-buoyant sediment transport in shallow-water
flows is illustrated by computing an erosional dam-break flow. This benchmark depicts the capabilities of the present model to account for erosional processes,
as well as to model the boundary between the traction carpet (or bed load layer) and the intermittently suspended sediment, which cannot be sharply defined
in hyperconcentrated flows.

1Work supported by the Spanish MICINN under project number DPI2008-06624-C03-02.

8:13AM MH.00002 Gravity currents in stratified fluids1, PAUL LINDEN, University of Cambridge — This paper
discusses the application of a model for the speed of a gravity current in an unstratified ambient fluid to the case where the ambient fluid is stratified with
a constant vertical density gradient. The model is based on the fact, first noted by Ungarish & Huppert (2002), that away from the front of the current the flow
is hydrostatic so that the driving pressure difference can be determined from a vertical integral of the density field. An energy-conserving model derived by
Shin et al. (2004) for a gravity current in an unstratified fluid is modified to take account of the changed pressure difference. This modified model is compared
with lock-release laboratory experiments of Maxworthy et al. (2002) and numerical simulations by Maxworthy et al. (2002), Birman et al. (2007) and White
& Helfrich (2008). Excellent agreement between the predicted and observed current speeds is found for supercritical currents for a wide ratio of dimensionless
lock-depths. Subcritical currents, on the other hand, are observed to travel faster than predicted by this model. The reasons for these behaviours are discussed
and the roles of the internal waves generated in the ambient stratification are evaluated.

1This research was supported by the National Science Foundation grant CTS - 0756396.

8:26AM MH.00003 Initial lock ratio effects on the dynamics of constant-volume density cur-
cents, THOMAS BONOMETTI, IMFT, Université de Toulouse, INPT, UPS, CNRS, MARIUS UNGARISH, Department of Computer Science, Technion,
Haifa, Israel, S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL, USA — The behaviour of
non-Boussinesq constant-volume density currents of density $\rho_c$, released from a lock of height $h_0$ and length $x_0$ into a ambient of height $H$ and density $\rho_a$, are
considered. Two-dimensional Navier-Stokes simulations are used to cover a wide range of density ratio $10^{-2} < \rho_c/\rho_a < 10^2$ and initial lock ratio $0.5 \leq \lambda \leq 18.75$.
The Navier-Stokes results are compared with predictions of a shallow-water model. In particular, we derive novel insights on the influence of the lock aspect
ratio $\lambda = x_0/h_0$ on the shape and motion of the current in the slumping stage. It is shown that a critical value exists, $\lambda_{crit}$, the dynamics of the current is
significantly influenced by $\lambda$ if below $\lambda_{crit}$. We conjecture that this critical lock ratio depends on two characteristic time scales, namely the slumping time and
the time of formation of the current’s head. Comparison with space-time diagrams obtained from the Navier-Stokes simulations show a good agreement.
We present a simple analytical model which support the observation that for a light current the speed of propagation is proportional to $\lambda^{1/4}$ when $\lambda < \lambda_{crit}$.

8:39AM MH.00004 The dam-break of non-Boussinesq gravity currents of various fractional
depth: two-layer shallow-water results, MARIUS UNGARISH, Technion Haifa Israel — The dam-break initial stage of propagation of a
current gravity released from a lock of length $x_0$ and height $h_0$ into an ambient fluid in a channel of height $H^+$ is considered. The system contains heavy
and light fluids, of density $\rho_H$ and $\rho_L$, respectively. When the Reynolds number is large, the resulting flow is governed by the parameters $R = \rho_L/\rho_H$ and
$H = H^+/h_0$. We focus attention on non-Boussinesq effects, when the parameter $R$ is not close to $1$, in this case significant differences appear between the
"light" (top) current and the "heavy" (bottom) current. Using a shallow-water two-layer formulations, we obtain "exact" analytical solutions for the thickness
and speed of the current and ambient by the method of characteristics. We shown that a jump (instead of a rarefaction wave) propagates into the reservoir
when $H < h_{crit}(R)$, and that propagation with critical speed occurs for some combinations of $H, R$. The theory is applied to the full-depth lock exchange
$H = 1$ problem, and also to more general cases $H > 1$. Comparisons to previously published results are discussed. This is a significant extension of the
Boussinesq problem (which is recovered by the present solution for $R = 1$), which elucidates the non- Boussinesq effects during the first stage of propagation
of lock-released gravity currents.
8:52AM MH.00005 Turbulent Mixing in Gravity Currents with Transverse Shear, BRIAN WHITE, UNC Chapel Hill, Marine Sciences, KARL HELFRICH, Woods Hole Oceanographic Institution, ALBERTO SCOTTI, UNC-Chapel Hill — A parallel flow with horizontal shear and density gradient undergoes an intensification of the shear by gravitational tilting and stretching, rapidly breaking down into turbulence. Such flows have the potential for substantial mixing in estuaries and the coastal ocean. We present high-resolution numerical results for the mixing efficiency of these flows, which can be viewed as gravity currents with transverse shear, and contrast them with the well-studied case of stably stratified, homogeneous turbulence (uniform vertical density and velocity gradients). For a sheared gravity current, the buoyancy flux, turbulent Reynolds stress, and dissipation are well out of equilibrium. The total kinetic energy first increases as potential energy is transferred to the gravity current, but rapidly decays once turbulence sets in. Despite the non-equilibrium regime where the dissipation rate is large compared with viscosity and stratification, $\epsilon/\omega^2 > 100$, further declining as turbulence decays and kinetic energy dissipation dominates the buoyancy flux. In general, the mixing rate, parameterized by a turbulent eddy diffusivity, increases with the strength of the transverse shear.

9:05AM MH.00006 Numerical Simulations of Two-layer Bores, LAURA BRANDT, KYLE BRUCKER, JAMES ROTTMAN, DOUG DOMMERMUTH, SAIC — Implicit LES is used for a systematic study of the energetics of upstream-propagating bores generated by trans-critical flow of two density layers over two-dimensional and three-dimensional topography. The results of these simulations are compared with laboratory measurements of $a$ free-surface elevation at several locations down the flume and $b$ impact pressure at the base of the breakwater. The free-surface elevations as predicted by NFA are in excellent agreement with the experimental measurements. This shows that NFA can simulate the propagation of waves over long distances with minimal amplitude and dispersion errors. Pressures that are induced by the jet are important because in certain coastal areas buildings must be designed to sustain tsunami loads. The pressure predictions over the duration of breaking agree very well with laboratory measurements. The peak pressures predicted by NFA are in excellent agreement with experiments.

9:18AM MH.00007 Numerical Prediction of Wave Forces on a Breakwater under Tsunami Loading, KYLE A. BRUCKER, SAIC, MARY BETH OSHNACK, Oregon State University, THOMAS T. O’SHEA, SAIC, DAN COX, Oregon State University, DOUGLAS G. DOMMERMUTH, SAIC — Numerical Flow Analysis (NFA) predictions of wave propagation and wave-impact loading are compared to the Oregon State University (OSU) O.H. Hinsdale Wave Research Laboratories Tsunami experiments (Oshnack, et al. 2009). The simulations were designed to replicate the experiments such that a soliton is sent down a wave flume, runs up a small beach, and impacts with a breakwater. The soliton is 1.2m high in a water depth of 2.29m and travels over 61m before hitting the breakwater. The NFA predictions are compared to laboratory measurements of a) free-surface elevation in several locations down the flume and b) impact pressure at the base of the breakwater. The free-surface elevations as predicted by NFA are in excellent agreement with the experimental measurements. This shows that NFA can simulate the propagation of waves over long distances with minimal amplitude and dispersion errors. Pressures that are induced by the jet are important because in certain coastal areas buildings must be designed to sustain tsunami loads. The pressure predictions over the duration of breaking agree very well with laboratory measurements. The peak pressures predicted by NFA are in excellent agreement with experiments.

9:31AM MH.00008 Alternative analysis of the temperature distribution in the interior ocean$^1$, SHENG-QI ZHOU, KAI LI, LING QU, South China Sea Institute of Oceanology, CAS, China, DEEP-SEA DYNAMICS TEAM — In the interior ocean (below the thermocline), temperature variation is less influenced by the wind Ekman upwelling and the surface heating, and it is mainly dominated by the balance between the downward mixing of heat by turbulence and the upward transport of heat by the vertical current. In 1966, Munk proposed one-dimension advection-diffusive model, and he found that the temperature and the depth has an exponential relationship when the upward speed and the vertical eddy diffusivity are assumed to be constant. We have analyzed the global ocean potential temperature in 2008 from the ARGOS delay model temperature dataset. It has been found that the exponential relationship model is not valid when the wave number is large. The results of these simulations are compared with the empirical law relationship has been found in various regions, such as in the North Pacific Ocean. The different relationships may suggest that the vertical eddy diffusivity has different dependencies on the depth, which may be useful to the global ocean models.

$^1$Work supported by the China NSF grant.

9:44AM MH.00009 Energy spectra of stably stratified turbulence, YOSHIKUMI KIMURA, Nagoya Univ., JACKSON HERRING, NCAR — We investigate energy spectra of stably stratified turbulence using direct numerical simulations (DNS) at a resolution of 1024$^3$. The calculation is done by solving the 3D Navier-Stokes equations under the Boussinesq approximation pseudo-spectrally. Using toroidal-poloidal decomposition (Craya-Herring decomposition), the velocity field is decomposed into the vortex mode and the wave mode. In general, both the wave and vortex spectra are consistent with a Kolgomorov-like $k^{−5/3}$ range at sufficiently large $k$. At large scales, and for sufficiently strong stratification the wave spectrum is a steeper $k^{−5/3}$, while that for the vortex component is consistent with $k^{−3}$. Here $k_\perp$ is the horizontally gathered wave numbers. In contrast to the horizontal wave number spectra, the vertical wave number spectra show very different features. We can observe clear $k_\perp^{3/2}$ dependence for small scales while the large scales show rather flat spectra. We link these spectra to the 2nd order structure functions of the velocity correlations in the horizontal and vertical directions. Finally we study the inviscid limit in which the highest wave-numbers are progressively thermalized, leaving the smaller wave numbers to adjust to their internal dynamics ansa dissipate. In this case, we see–for the non-thermalized components–similar dynamics as that for the finite Reynolds case.

9:57AM MH.00010 Numerical simulations of the transport of passive scalars around obstacles in tidal flows, HYEYUN KU$^1$, SUBHAS VENAYAGAMOORTHY, Colorado State University — This research is centered on understanding the mixing and transport of passive scalars around obstacles with and without drag in tidal flows. High-resolution two- and three-dimensional numerical simulations were performed of a passive scalar in an idealized domain to study the effects of drag and different flow conditions (tides and currents) on the evolution of a passive scalar. The horizontal dispersion coefficient is quantified as a function of three non-dimensional parameters namely: the drag coefficient, $C_D$ (imparted by the obstacle); the ratio of the tidal to mean flow velocity amplitudes, $U_T/U_M$; and an oscillatory tidal excursion length parameter, $2U_T/\omega D$, where $\omega$ is the frequency and $D$ is the diameter of an obstacle. The drag exerted by a porous obstacle blocks the flow partially and causes the deceleration of the flow, the shedding of vortices and the formation of a downstream wake. Results of the scalar field with and without drag for both uni-directional and oscillatory flow fields are presented. The simulation results highlight the complex dispersion patterns around submerged obstacles and provide an understanding on pollutant dispersion in the atmosphere such as urban cities and in water bodies such as the coastal ocean where vegetation tends to obstruct flow.

$^1$Graduate Student

Tuesday, November 23, 2010 8:00AM - 9:57AM – Session MJ Acoustics III: Oscillatory Excitations Long Beach Convention Center 201A
The effect of drop oscillations on the scavenging of solid particles is studied using an ultrasonic transducer to levitate a water droplet in an airflow of particles. Shape mode oscillations are induced in the droplet by modulating the acoustic field used for levitation. The effect of oscillation frequency, the oscillation amplitude, and the drop diameter on the scavenging of particles is presented. The particle diameters are on the order of 1 \( \mu \text{m} \) and the drop diameters are on the order of 1 mm. Although single droplets are studied here, the application of interest is improved scavenging of particles by spray drops. Specifically, improving the elimination of coal dust particles from mines using waters sprays excited ultrasonically is of interest.

Support from the Center for Disease Control is gratefully acknowledged.

8:13AM MJ.00002 Droplet actuation by surface acoustic waves: an interplay between acoustic streaming and radiation pressure, PHILIPPE BRUNET, IEMN, CNRS - UMR 8520, MICHAEL BAUDOIN, IEMN, CNRS - UMR 8520 & Universite Lille 1, OLIVIER BOU MATAR, IEMN, CNRS - UMR 8520 & Ecole Centrale de Lille, FARZAM ZOUESHTIAGH, IEMN, CNRS - UMR 8520 & Universite Lille 1, AIMAB/FILMS TEAM — Surfaces acoustic waves (SAW) are known to be a versatile technique for the actuation of sessile drops. Droplet displacement, internal mixing or drop splitting, are amongst the elementary operations that SAW can achieve, which are useful on lab-on-chip microfluidics benches. On the purpose to understand the underlying physical mechanisms involved during these operations, we study experimentally the droplet dynamics varying different physical parameters. Here in particular, the influence of liquid viscosity and acoustic frequency is investigated: it is indeed predicted that both quantities should play a role in the acoustic-hydrodynamic coupling involved in the dynamics. The key point is to compare the relative magnitude of the attenuation length, i.e. the scale within which the acoustic wave decays in the fluid, and the size of the drop. This relative magnitude governs the relative importance of acoustic streaming and acoustic radiation pressure, which are both involved in the droplet dynamics.

8:26AM MJ.00003 Finite-Element Modeling of Forced Axisymmetric Sessile Drop Oscillation Using a Moving Mesh, CHRIS FORSTER, MARC SMITH, ARI GLEZER, Georgia Institute of Technology — The behavior of a sessile drop undergoing axisymmetric oscillations is studied using the finite element method with a moving mesh to track the fluid interface. The drop is modeled in two different ways. In the first model, the surrounding fluid is assumed to have negligible viscosity and inertia. The drop motion is computed using a single fluid domain for the drop and solving the incompressible continuity and Navier-Stokes equations with appropriate interfacial boundary conditions. In the second model, the surrounding fluid is explicitly modeled. Here, two fluid domains are used, each having their own respective set of continuity and Navier-Stokes equations and these are fully coupled with the appropriate interfacial boundary conditions. The behavior of the drop is investigated for both pinned and dynamic contact line boundary conditions. A comparison of results from the two models and to the results from previous work is offered. The focus of this work is on extending this simulation to higher modes of oscillation where the effects of viscous damping from both the primary and surrounding fluids are more important.

Support by the Office of Naval Research.

8:39AM MJ.00004 Mechanically-Excited Sessile Drops, CHUN-TI CHANG, JOSHUA BOSTWICK, SUSAN DANIEL, PAUL STEEN, Cornell University — The volume and contact-line mobility of a sessile drop determine the frequency response of the drop to mechanical excitation. A useful signature of the drop is its response to a sweep of frequency. At particular frequencies the drop exhibits standing wave patterns of different mode numbers and/or azimuthal, spinning motion. We report observations of the spectrum of standing wave patterns and compare to predictions of a linear stability theory. On the side of application, the results suggest how to tune the pinning-unpinning of a sessile drop in order to maximize its translation.

8:52AM MJ.00005 Compressibility and Related Thermal and Diffusional Effects in Acoustics Streaming, SATWINDAR SADHAL, University of Southern California, ALEXEY REDNIKOV, Université Libre de Bruxelles — The effects of air compressibility have been analyzed for acoustic streaming due to acoustic fields interacting with solid boundaries. These include possible influence of the thermal boundary layer and, if the surrounding gas is a multicomponent mixture, of the diffusional boundary layer. It is well known that acoustic streaming originates in the viscous boundary layer at the surface of a particle, where flow is genuinely rotational and thus, a steady flow component is generated as soon as nonlinearities are taken into account. While this crucial role of the viscous boundary layer has been widely recognized, little attention has been paid to the role of the thermal boundary. The latter one, besides being of interest to purely thermoacoustics, may also affect the streaming itself. Since density is a function of temperature (as well as gas composition), a sharp temperature variation in the boundary layer gives rise to the corresponding density variation, influencing the flow field (continuity equation), and globally affecting streaming intensity.

9:05AM MJ.00006 Numerical simulation of particle dispersion in an acoustic field, J. CLECKLER, F. LIU, S. ELGHOBASHI, University of California, Irvine — Particles with small relaxation time, \( \tau_p \), subjected to sound waves for many acoustic periods exhibit both periodic motion and mean drift. Particle acceleration in an acoustic flow field is often modeled via a linearized Stokes drag law. This simple model can predict the oscillatory particle velocity amplitude for large particle-to-fluid density ratios, \( \rho_p/\rho_f \), and small velocity-amplitude acoustic waves. However, this model is not accurate for other conditions and does not predict particle drift velocities. We present the results of two-dimensional numerical simulations in which the particle trajectories are obtained via the complete Lagrangian particle motion equation which includes the forces due to nonlinear Stokes drag, Basset’s unsteady viscous drag, pressure gradient, virtual mass and gravity. Particle behavior is found to depend on three non-dimensional parameters: \( (\rho_p/\rho_f, \omega \tau_p, M) \), where \( \omega \) is the acoustic frequency, and the Mach number, \( M \), which is the ratio of the acoustic wave velocity amplitude to the speed of sound. Results for large \( \rho_p/\rho_f \) are in good agreement with the experimental results of Gonzalez et al. (2000) for the range of frequencies tested. Results for other conditions agree with a perturbation solution of the Lagrangian particle motion equation for moderate strength acoustic waves. Particle model simplifications are recommended for important ranges of the three parameters, \( (\rho_p/\rho_f, \omega \tau_p, M) \).

9:18AM MJ.00007 Vortex structures and heat transfer in acoustic streaming flows, IN MEI SOU, JOHN ALLEN, CHRISTOPHER LAYMAN, CHITTARANJAN RAY, University of Hawaii at Manoa — The velocity and heating in an acoustically induced streaming flow are investigated using simultaneous particle image velocitymetry (PIV) and infrared thermography. This study is motivated by the increasing applications of ultrasound-based processing in various fields such as wastewater treatment and biotechnology. The characterization of the acoustic streaming field is an important step in the overall design for sonochemical reactors used in these treatment processes. Results of the coherent structures and heating are obtained from an experimental study of acoustic streaming in a clear acrylic tank. The PIV resolved velocity fields show a jet-like flow along the centerline of the horn and a main vortex pair propagating in the direction of the flow. The coherent vortex structures are examined in terms of the swirling strength and Lagrangian coherent structures (LCS). The swirling strength is used to visualize the vortices in the Eulerian reference frame while the LCS approach is used to reveal the underlying flow structures for the unsteady case. The swirling strength is defined as the imaginary part of the complex eigenvalue of the local velocity gradient tensor. The LCS is defined as the local maxima of the finite-time Lyapunov exponent (FTLE). We present the evolution of the temperature fields together with the corresponding swirling strength and the LCS calculations.
9:31AM MJ.00008 Radiative heating of acoustically levitated nano-silica droplets: Internal flow pattern leading to ring or bowl shaped structure  . ABHISHEK SAHA, ERICK TIJERINO, RANGANATHAN KUMAR, University of Central Florida, SAPTARSHI BASU, Indian Institute of Science — An experimental setup using radiative heating has been used to understand the thermophysical phenomena inside acoustically levitated droplets. In this transformation process, through IR thermography and high speed imaging, events such as vaporization, precipitation have been recorded at high temporal resolution; leading to ring or bowl shaped structures. High solute loading is seen to form high concentration precipitate near with a weak center linkage which results in a horizontal ring formation initially. Droplet recirculation is more effective at lower concentrations, inducing a bridge formation near the center leading to a bowl formation. With non-uniform particle distribution, these structures can experience rupture which modifies the droplet rotational speed with preferential orientation. PIV on sub millimeter sized droplets shows presence of strong single core vortex around droplet center. Study with droplet diameter and viscosity of the liquid leads to the conclusion that the strength of the vortex is dependent on these parameters. Further investigation with LIF confirms preferential accumulation of particles at the bottom of the droplet.

9:44AM MJ.00009 Effectiveness of Thin Urethane Coatings for Attenuating Flow Noise in SONAR Arrays  . WILLIAM KEITH, NAVSEA Newport, IAN COOK, Cornell Univ, ALIA FOLEY, KIMBERLY CIPOLLA, NAVSEA Newport — The wall pressure fluctuations exerted on the wall beneath a turbulent boundary layer introduce flow noise which limits the performance of SONAR arrays. One method for mitigating this flow noise is to increase the standoff distance between the fluid-solid interface and the sensors in the array. A parametric study was conducted comparing the energy spectra of wall pressure fluctuations measured with wall pressure sensors under .025" and .05" coatings of urethane bonded with an aqueous, flat plate turbulent boundary layer. Measurements made with the sensors flush mounted in a flat plate, directly under the boundary layer (no coating) serve as a baseline for comparison. Reynolds number effects are considered, as is the effectiveness of boundary layer variables for collapsing the data (i.e. scaling relationships).

Tuesday, November 23, 2010 8:00AM-10:10AM
Session MK Biofluids: Cellular II  Long Beach Convention Center 201B

8:00AM MK.00001 Rheology of capsule suspension  . PROSENJIT BAGCHI, Rutgers University, R. MURTHY KALLURI — Rheology of suspension of liquid-filled elastic capsules in linear shear flow is studied from three-dimensional numerical simulations using a front-tracking method. First, we consider dilute suspension capsules of spherical resting shape for which only a steady tank- treading motion is observed. We find a novel result that the capsule suspension exhibits a shear viscosity minimum at moderate values of the viscosity ratio, and high capillary numbers. The shear viscosity minimum exists for capsules with area-dilating membranes, but not for those with nearly-incompressible membranes. Physical mechanisms underlying these results are studied by decomposing the particle stress tensor into a contribution due to the elastic stresses in the capsule membrane, and a contribution due to the viscosity differences between the internal and suspending fluid. It is shown that the elastic contribution is shear-thinning, while the viscous contribution is shear-thickening. We then consider dilute suspension of oblate shape capsules which undergo unsteady motion such as swinging and tumbling. The effect of such unsteady dynamics on the time-dependent rheology is addressed. Finally, we consider dense suspension, and observe that the shear viscosity minimum disappears with increasing capsule volume fraction.

8:13AM MK.00002 Capsule deformation and orientation in general linear flows  . ALEX SZATMARY, CHARLES EGGLETON, University of Maryland, Baltimore County — We considered the response of spherical and non-spherical capsules to general flows. (A capsule is an elastic membrane enclosing a fluid, immersed in fluid.) First, we establish that nonspherical capsules align with the imposed irrotational linear flow; this means that initial orientation does not affect steady-state capsule deformation, so this steady-state deformation can be determined entirely by the capillary number and the type of flow. The type of flow is characterized by \( r = 0 \) for axisymmetric flows, and \( r = 1 \) for planar flows; intermediate values of \( r \) are combinations of planar and axisymmetric flow. By varying the capillary number and \( r \), all irrotational linear flow Strikkes flows can be generated. For the same capillary number, planar flows lead to more deformation than uniaxial or biaxial extensional flows. Deformation varies monotonically with \( r \), so one can determine bounds on capsule deformation in general flow by only looking at uniaxial, biaxial, and planar flow. These results are applicable to spheres in all linear flows and to ellipsoids in irrotational linear flow.

8:26AM MK.00003 An integrated model of microtubule-based pronuclear motion in the single-celled C. elegans embryo  . TAMAR SHINAR, MICHAEL SHELLEY, Courant Institute, NYU — We present an integrated computational model of microtubule-based pronuclear motion in the single-celled C. elegans embryo. In this model, centrosomes initiate stochastic microtubule growth and these microtubules interact with motor proteins distributed in the cytoplasm. Consequent pulling forces drag the pronucleus through the cytoplasm, here modeled as an incompressible, Newtonian fluid whose motions are constrained by contact with the cell periphery. The cell periphery also limits microtubule growth. Our computational method is based on an immersed boundary formulation which allows for the simultaneous treatment of fluid flow and the dynamics of structures immersed within. Our simulations show pronuclear migration, and moreover, a geometry-dependent pronuclear centration and rotation very similar to that observed in vivo. We study the dynamic interaction of motor proteins embedded in the fluid with microtubule filaments, allowing for relative motion of fluid along MT tracks as has been observed experimentally. We demonstrate numerically that this is sufficient to propel the pronucleus while causing a counterflow of the cytoplasm.

8:39AM MK.00004 Monte Carlo Simulations of Absolute Binding Free Energy of Targeted Nanocarriers to Cell Surfaces  . JIN LIU, B. ZERN, P.S. AYYASWAMY, D.M. ECKMANN, V.R. MUZYKANTOV, R. RADHAKRISHNAN, University of Pennsylvania — We have developed a computational methodology based on Metropolis Monte Carlo and the weighted histogram analysis method (WHAM) to calculate the absolute binding free energy between functionalized nanocarriers (NC) and endothelial cell (EC) surfaces. The calculated binding affinities agree quantitatively with the measurements of specific antibody coated NCs (100 nm in diameter) to intracellular adhesion molecule-1 (ICAM-1) expressing EC surface in in vitro experiments. We then systematically explore the effects of experimentally tunable parameters including the antibody surface coverage \( \sigma \), NC, glycocalyx, shear flow and NC size. Of particular biological significance, our model predicts a threshold \( \sigma \) value below which the NC binding affinities reduce drastically and drop below that of single anti-ICAM-1 molecule to ICAM-1; our results reveal that this is due to a change in the multivalency (or number of bonds formed per NC). This trend and threshold value are recovered exactly in the in vivo measurements of the endothelium targeting of NCs in the pulmonary vascular in mice.

3Supported by NIH: R01-EB006818, R01-HL087036, NSF/CBET: 0853389.

8:52AM MK.00005 Measurement of Cytoplasmic Streaming in Drosophila Melanogaster  . SUJOY GANGULY, LUCY WILLIAMS, ISABEL PALACIOS, RAYMOND GOLDSTEIN, University of Cambridge — During stage 9 of Drosophila melanogaste oogenesis flow of the oocyte cytoplasm, driven by kinesin 1 motor protein is observed. This cytoplasmic streaming is analyzed by PIV in both wild type and kinesin light chain mutants, revealing striking statistical differences. Further measurements of the rheology of the oocyte allow for estimations of the mechanical energy needed to generate the observed flows.
9:05AM MK.00006 Microrheology Using Optical Tweezers at the Air-Water Interface1. THOMAS BOATWRIGHT, University of California, Irvine. ALEX LÉVINE, University of California, Los Angeles, MICHAEL DENNIN, University of California, Irvine — Microrheological techniques have been used successfully to determine mechanical properties of materials important in cellular structure. Also critical to cellular mechanical functions are biological membranes. Many aspects of biological membranes can be modeled using Langmuir monolayers, which are single layers of surfactants at the air-water interface. The macroscopic mechanical properties of Langmuir monolayers have been extensively characterized. In contrast to macroscopic measurements, we report on experimental methods for studying the rheological properties of Langmuir monolayers on the micron scale. A water immersion optical tweezers system is used to trap ~1 micron diameter beads in a monolayer. The passive motion of the trapped beads is recorded at high frequency and the complex shear modulus is calculated. Preliminary microrheological data of a fatty acid monolayer showing dependence on surface pressure will be presented. Experimental obstacles will also be discussed.

1 NSFDMR-0907212 and the Research Corporation

9:18AM MK.00007 Protein amyloid formation: Effects of shear, advection and interfaces. DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute — The aggregation of proteins into amyloid assemblies, which is associated with diseases such as Alzheimer’s, is characterized by the unfolding of a given protein from its native state, the aggregation of some of these denatured species into nuclei, and further elongation into fibrils from these precursors. Previous observations have shown that shearing of the protein solution has a significant effect on the aggregation kinetics, but a clear understanding of the separate effects of shear forces, convective transport of species, and interfaces on the amyloid formation process is yet to be established. In the present work, we consider various shearing flow geometries and boundary conditions (e.g. gas/liquid and solid/liquid interfaces). The kinetics of the process are followed in time by measuring the change of protein in solution, and by microscopic observation of the aggregated species at the interfaces and in the bulk.

9:31AM MK.00008 Modelling Protein-induced Membrane Deformation using Monte Carlo and Langevin Dynamics Simulations1, R. RADHAKRISHNAN, N. AGRAWAL, University of Pennsylvania, N. RAMAKRISHNAN, P.B. SUNIL KUMAR, Indian Institute of Technology Madras, J. LIU, University of Pennsylvania — In eukaryotic cells, internalization of extracellular cargo via the cellular process of endocytosis is orchestrated by a variety of proteins, many of which are implicated in membrane deformation/bending. We model the energetics of deformations techniques by using the Helfrich Hamiltonian using two different formalisms: (i) Cartesian or Monge Gauge using Langevin dynamics; (ii) Curvilinear coordinate system using Monte Carlo (MC). Monge gauge approach which has been extensively studied is limited to small deformations of the membrane and cannot describe extreme deformations. Curvilinear coordinate approach can handle large deformation limits as well as finite-temperature membrane fluctuations; here we employ an unstructured triangular mesh to compute the local curvature tensor, and we evolve the membrane surface using a MC method. In our application, we compare the two approaches (i and ii above) to study how the spatial assembly of curvature inducing proteins leads to vesicle budding from a planar membrane. We also quantify how the curvature field of the membrane impacts the spatial segregation of proteins.

1 Supported by NSF/CBET:0853389 and NIH: R01-EB006818.

9:44AM MK.00009 Effect of Reynolds number on 2-D protein crystallization at the air/water interface. JAMES YOUNG, DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — X-ray diffraction is the primary technique used to obtain a detailed description of a protein on the molecular level and as such, has yielded essential information about protein structures and protein-ligand interactions. However a major drawback of this technique is that the protein must first be crystallized which is often a very difficult and inefficient process. It has been shown previously that the process of two-dimensional protein crystallization on lipid monolayers at the air/water interface can be enhanced by a shearing flow. Here we examine the relationship between Reynolds number and the crystal growth process using the deep-channel surface viscometer geometry, which consists of an annular region bounded by stationary inner and outer cylinders and driven by a constant rotation of the floor. The interfacial velocity measurements are compared to Navier-Stokes computations with the Boussinesq-Screen surface model.

9:57AM MK.00010 Quantification of Electroporation-Mediated Propidium Iodide Delivery into 3T3 Cells. MOHAMED M. SADIK, JIANBO LI, JERRY W. SHAN, DAVID I. SHREIBER, HAO LIN, Rutgers University — Electroporation is an effective means to deliver exogenous molecules into the cellular cytoplasm, while simultaneously maintaining cell viability and functionality. In this technique, an applied electric field transiently permeabilizes the cellular membrane to enable molecular exchange. The main objective of the current work is to identify the transport mechanisms involved during electroporation, and to quantify the amount of molecules delivered into the cellular cytoplasm. An optical diagnostic system is developed to examine the transport of Propidium Iodide (PI) into 3T3 mouse fibroblast cells. Upon entering the permeabilized cell, PI binds to DNA/RNA within the cytoplasm to emit fluorescence, which is measured to track the dynamic accumulation of the dye within the cell. The results show that the total fluorescence intensity increases with a decreasing buffer electrical conductivity. The data are compared with numerical simulations, which reveals good agreement. The experimental observations and numerical analysis demonstrate that: 1) Electrophoresis plays a dominant role in mediating the transport. 2) The transport mechanisms involved during electroporation, and to quantify the amount of molecules delivered into the cellular cytoplasm. An optical diagnostic system is developed to examine the transport of Propidium Iodide (PI) into 3T3 mouse fibroblast cells. Upon entering the permeabilized cell, PI binds to DNA/RNA within the cytoplasm to emit fluorescence, which is measured to track the dynamic accumulation of the dye within the cell. The results show that the total fluorescence intensity increases with a decreasing buffer electrical conductivity. The data are compared with numerical simulations, which reveals good agreement. The experimental observations and numerical analysis demonstrate that: 1) Electrophoresis plays a dominant role in mediating the transport. 2) The transport mechanisms involved during electroporation, and to quantify the amount of molecules delivered into the cellular cytoplasm. An optical diagnostic system is developed to examine the transport of Propidium Iodide (PI) into 3T3 mouse fibroblast cells. Upon entering the permeabilized cell, PI binds to DNA/RNA within the cytoplasm to emit fluorescence, which is measured to track the dynamic accumulation of the dye within the cell. The results show that the total fluorescence intensity increases with a decreasing buffer electrical conductivity. The data are compared with numerical simulations, which reveals good agreement. The experimental observations and numerical analysis demonstrate that: 1) Electrophoresis plays a dominant role in mediating the transport. 2) An electrokinetic mechanism, field-amplified sample stacking, controls the achievable delivery efficiency. The study in this work is an important step toward the quantification as well as the eventual improvement of this useful technique.

Tuesday, November 23, 2010 8:00AM - 10:10AM –
Session ML Biofluids: Physiological Cardiovascular I  Long Beach Convention Center 202A

8:00AM ML.00001 High resolution simulation of the left heart hemodynamics in patient-specific anatomies. TRUNG LE, IMAN BORAZJANI, FOTIS SOTIROPOULOS, University of Minnesota — Understanding left-ventricle (LV) hemodynamics is critical prerequisite for developing new methods for diagnosing, treating and managing left heart dysfunction diseases. We develop a high resolution computational model of the left heart based on data from MRI scan images from a healthy volunteer and develop a physiologic, cell-activation based myocardial model for calculating the kinematics and the mechanical response of mechanical valve leaflets. The computed results show that the LV motion resulting from the model gives rise to global agreement. The experimental observations and numerical analysis demonstrate that: 1) Electrophoresis plays a dominant role in mediating the transport. 2) An electrokinetic mechanism, field-amplified sample stacking, controls the achievable delivery efficiency. The study in this work is an important step toward the quantification as well as the eventual improvement of this useful technique.

Session ML Biofluids: Physiological Cardiovascular I  Long Beach Convention Center 202A
8:13AM ML.00002 Aortic Wave Dynamics and Its Influence on Left Ventricular Workload. NIEMA PAHELEVEN, MORTEZA GHARIB, California Institute of Technology — Clinical and epidemiologic studies have shown that hypertension plays a key role in development of left ventricular (LV) hypertrophy and ultimately heart failure mostly due to increased LV workload. Therefore, it is crucial to diagnose and treat abnormal high LV workload at early stages. The pumping mechanism of the heart is pulsatile, thus it sends pressure and flow wave into the compliant aorta. The wave dynamics in the aorta is dominated by interplay of heart rate (HR), aortic rigidity, and location of reflection sites. We hypothesized that for a fixed cardiac output (CO) and peripheral resistance (PR), interplay of HR and aortic compliance can create conditions that minimize LV power requirement.

We used a computational approach to test our hypothesis. Finite element method with direct coupling method of fluid-structure interaction (FSI) was used. Blood was assumed to be incompressible Newtonian fluid and aortic wall was considered elastic isotropic. Simulations were performed for various heart rates and aortic rigidities while inflow wave, CO, and PR were kept constant. For any aortic compliance, LV power requirement becomes minimal at a specific heart rate. The minimum shifts to higher heart rates as aortic rigidity increases.

8:26AM ML.00003 Leveraging theory from cosmodynamics to study the the effect of the pulsating heart on coronary arteries in a large-scale Lattice Boltzmann simulation. Amanda Peters, Harvard University, Simone Melchionna, Jonas Latt, EPFL, Săuro Succi, Consiglio Nazionale delle Ricerche, Efthimios Kaxiras, Harvard University — We present a computational method for the simulation of cardiovascular flows in realistic human geometries derived from computed tomography angiography (CTA) data. The simulation is based on the Lattice Boltzmann method to model the blood flow in large-scale arterial systems and extends previously published studies using static geometries to include the effect of the pulsating heart on the coronary arteries. We provide here the derivation for introducing the deformational forces exerted on the arterial flows from the movement of the heart by borrowing concepts from cosmodynamics. The deformational forces are then cast into the kinetic formalism by using a Gauss-Hermite projection procedure. In this presentation, we will discuss this method as well as provide an analysis of the impact of these additional forces on the endothelial shear stress, a quantity associated with the localization and progression of heart diseases like atherosclerosis.

8:39AM ML.00004 Analysis of velocity fluctuations downstream of a bileaflet mechanical heart valve. Marcio Forleo, Lakshmi Dasí, Colorado State University — Bileaflet mechanical heart valves are widely used to replace diseased aortic heart valves. The stresses induced by the rich and unsteady non-physiological flow structures have been the focus to evaluate red blood cells damage and platelet activation, develop flow control strategies, or improve valve designs. In this study, we analyzed the flow fields obtained downstream of a bileaflet mechanical heart valve using time-resolved particle image velocimetry under pulsatile and steady flow conditions. Our study demonstrates the rich dynamics downstream of the valve and weighs the relevance of unsteady effects vs inertia effects on the different flow structures. Power spectrum analyses of the turbulent fluctuations highlight the highly anisotropic influence and the limited applicability of classical self-similar turbulence theory in describing the small-scale structures in the immediate vicinity of the valve.

8:52AM ML.00005 Mitigation of Shear-Induced Blood Damage by Mechanical Bileaflet Heart Valves. Boris Zakharin, SiVakkumar Arjunon, Neelakantan Saikrishnan, Ajit Yogathan, Arie Glezer, Georgia Institute of Technology — The strong transient shear stress generated during the time-periodic closing of bileaflet mechanical heart valves that is associated with the formation of counter-rotating vortices near the leaflet edges may be damaging to blood elements and may result in platelet activation and therefore thrombosis and thromboembolism complications. These flow transients are investigated using fluorescent PIV in a new, low-volume test setup that reproduces the pulsatile physiological conditions associated with a 25 mm St. Jude Medical valve. The flow transients are partially suppressed and the platelet activation is minimized using miniature vortex generator arrays that are embedded on the surface of the leaflets. Measurements of the ensuing flow taken phase-locked to the leaflet motion demonstrate substantial modification of the transient vertical structures and concomitant reduction of Reynolds shear stresses. Human blood experiments validated the effectiveness of miniature vortex generators in reducing thrombus formation by over 42 percent.

9:05AM ML.00006 Visualization of Simulated Endothelial Shear Stress and Blood Flow in Coronary Arteries. Michelle Borkin, Harvard University, Charles L. Feldman, Brigham and Women’s Hospital, Harvard Medical School, Hanspeter Pfister, Harvard University, Simone Melchionna, Swiss Federal Institute of Technology, EPFL, Efthimios Kaxiras, Harvard University — Low endothelial shear stress (ESS) identifies areas of atherosclerotic disease lesion formation in the coronary arteries. However, it is impossible to directly measure ESS in vivo for an entire arterial tree. As part of the Multiscale Hemodynamics Project, computed tomography angiography (CTA) data is being used to obtain patient specific heart and coronary system geometries and then MUPHY, a multi-physics and multi-scale simulation code combining microscopic Molecular Dynamics (MD) with a hydro-kinetic Lattice Boltzmann (LB) method, is applied in order to simulate blood flow through the coronary arteries. Having effective visualizations of the simulation’s multidimensional output, including ESS, is vital for the quick and thorough non-invasive evaluation of the patient. To this end, we have developed new visualization tools and techniques to make the simulation’s output useful in a clinical diagnostic setting, examined the effectiveness of 2D versus 3D representations, and explored blood flow representations. The visualization methods developed are also applicable to other areas of fluid dynamics.

9:18AM ML.00007 Hemodynamic simulations in coronary aneurysms of a patient with Kawasaki Disease. Dibyendu Sengupta, Alisson Marsden, Mechanical and Aerospace Engineering Dept. UCSD, Jane Burns, Pediatrics Dept, UCSD — Kawasaki Disease is the leading cause of acquired pediatric heart disease, and can cause large coronary artery aneurysms in untreated cases. A simulation case study has been performed for a 10-year-old male patient with coronary aneurysms. Specialized coronary boundary conditions along with a lumped parameter heart model mimic the interactions between the ventricles and the coronary arteries, achieving physiologic pressure and flow waveforms. Results show persistent low shear stress in the aneurismal regions, and abnormally high shear at the aneurysm neck. Correlation functions have been derived to compare wall shear stress and wall shear stress gradients with recirculation time with the idea of localizing zones of calcification and thrombosis. Results are compared with those of an artificially created normal coronary geometry for the same patient. The long-term goal of this work is to develop links between hemodynamics and thrombotic risk to assist in clinical decision-making.

9:31AM ML.00008 ABSTRACT WITHDRAWN —
9:44AM ML.00009 A Mechanical System to Reproduce Cardiovascular Flows1, THOMAS LINDSEY, PIETRO VALSECHI, ExxonMobil Upstream Research Company, PUMPS AND PIPES INITIATIVE TEAM — Within the framework of the “Pumps&Pipes” collaboration between ExxonMobil Upstream Research Company and The DeBakey Heart and Vascular Center in Houston, a hydraulic control system was developed to accurately simulate general cardiovascular flows. The final goal of the development of the apparatus was the reproduction of the periodic flow of blood through the heart cavity with the capability of varying frequency and amplitude, as well as designing the systolic/diastolic volumetric profile over one period. The system consists of a computer-controlled linear actuator that drives hydraulic fluid in a closed loop to a secondary hydraulic cylinder. The test section of the apparatus is located inside a MRI machine, and the closed loop serves to physically separate all metal moving parts (control system and actuator cylinder) from the MRI-compatible pieces. The secondary cylinder is composed of nonmetallic elements and directly drives the test section circulatory flow loop. The circulatory loop consists of nonmetallic parts and several types of Newtonian and non-Newtonian fluids, which model the behavior of blood. This design allows for a periodic flow of blood-like fluid pushed through a modeled heart cavity capable of replicating any healthy heart condition as well as simulating anomalous conditions. The behavior of the flow inside the heart can thus be visualized by MRI techniques.

1 A collaboration between ExxonMobil URC and Methodist DeBakey Heart & Vascular Center.

9:57AM ML.00010 Investigating the Flow and Biomechanics of the Embryonic Zebrafish Heart, BRENNAN JOHNSON, DEBORAH CARRITY, LAKSHMI DASI, Colorado State University, CARDIOVASCULAR AND BIOPHYSICS LAB, CSU TEAM — Understanding flow and kinematic characteristics of the embryonic heart is a prerequisite to devise early intervention or detection methods in the context of congenital heart defects. In this study, the kinematics and fluid dynamics of the embryonic zebrafish heart were analyzed through the early stages of cardiac development (24–48 hours post-fertilization) in vivo using optical microscopy and high-speed video. Endocardial walls and individual blood cells were segmented from raw images and were tracked through the cardiac cycle. Particle tracking velocimetry analysis yielded quantitative blood cell velocity field, chamber volume, and flow rate information. It was seen that the pumping mechanism starts as a combined peristaltic and suction pump while the heart is in the tube configuration and transforms into a positive displacement pump after cardiac looping. Strong two-phase nature of the fluid is evident. This work provides us new understanding of the spatio-temporal characteristics of kinematics and blood cell velocity field inside the developing heart.

Tuesday, November 23, 2010 8:00AM - 10:10AM – Session MM Microfluids: General VII Long Beach Convention Center 202B

8:00AM MM.00001 Dynamics of desalination shocks in microstructures, ALI MANI, Stanford University, MARTIN BAZANT, MIT — We describe a nonlinear regime of desalination shocks that results from coupled conduction effects of electric double layers (surface conduction) in electrokinetic systems. Mani, Zangle, and Santiago (Langmuir, 25, 3898–3916) recently showed that sharp concentration gradients can be formed and propagate away from a microchannel/nanochannel junction, analogous to shock waves in gases. Propagation of these shocks in microchannels leaves behind a region with orders of magnitude lower salt concentration acting to desalinate the bulk electrolyte. In this talk we describe the basic dynamics of desalination and present the mathematical theory of shock existence and propagation in complex microstructures. We predict that desalination shocks accelerate and sharpen in narrowing structures and decelerate and weaken, even disappear, in widening channels. We will also discuss mathematical models for propagation of desalination shocks in porous media.

8:13AM MM.00002 Microfluidic flow characterized by light-induced local viscosity distribution, MASAHIRO MOTOSUKE, SHINJI HONAMI, Tokyo University of Science — Dominance of the viscous force in fluid flow is unique and important in a microfluidic field. The generation of property distribution, especially in viscosity, can induce the spontaneous change in flow structure. In this study, the effect of local variation of viscosity based on photothermal phenomena on the flow behavior in microchannel is examined. A light absorption in liquid causes a local change of the temperature, and it induces corresponding change in the viscosity because of the high temperature sensitivity in liquid viscosity. The flow velocity measurement of the stable liquid flow in microchannel with the photothermally-induced viscosity variation is performed by micro-PIV technique. As a result, flow structure around the hot spot is changed by the local property variation. The origin of the change in flow behavior is investigated numerically, and it is confirmed that only the viscosity has significant effect on the fluid flow in the small domain. Additionally, optimal profile of focused light irradiation to induce the significant change in flow field is obtained.

8:26AM MM.00003 Irreversible gelation of wormlike micelle solutions under microfluidic flow, PERRY CHEUNG, JOSHUA CARDELI, NEVILLE DUBASH, AMY SHEN, University of Washington — The formation of flow-induced gel-like structures in surfactant solutions containing wormlike micelles have previously been observed in macroscopic flow under applied shear in dilute solutions of cetyltrimethylammonium bromide (CTAB) and sodium salicylate (NaSal). However, the observed gelation phase transition is short-lived once the applied flow is stopped and reversibly disappears. Recently, irreversible gelation was achieved by applying high shear and extensional flows within a packed bed of microbeads in a microfluidic device [1]. We present here a further investigation of the irreversible flow-induced gelation of dilute solutions of CTAB/NaSal in microfluidic devices with microfabricated arrays of microchannels with varying post diameters and inter-post spacing. The onset of gelation at various surfactant concentrations and flow rates (both shear and extension rate) will be examined to determine the extent of this phenomenon.


8:39AM MM.00004 Temporally resolved 3D3C velocity measurements using confocal volumetric scanning, STEVEN KLEIN, Mechanical Engineering, Arizona State University, JONATHAN POSNER, Mechanical Engineering, Chemical Engineering, Arizona State University — A diagnostic platform for measuring three dimensional velocity fields in whole macroscopic volumes is presented. The imaging system is based on Nipkow spinning disk confocal microscopy. The confocal system provides optical sectioning using pinhole spatial filtering which rejects light originating from out of focus objects. Volumetric scanning is obtained by rapid translation of the high numerical aperture objective along a piezo stage. High speed optical sectioning and volumetric scanning of microscopic volumes can be used for real time visualization and velocimetry of three dimensional micro flows in applications such as 3D3C particle tracking velocimetry (PTV) and volumetric quantitative fluorescence imaging. Temporally resolved 3D3C velocity measurements of microchannel flow are presented at near video rates (10-20 Hz) using the scanning confocal system. Little post processing is required because only a single objective is used and no complex algorithms are needed to recover the depthwise velocity component nor to reconstruct the particle images.

8:52AM MM.00005 Self-consistent unstirred layers in osmotically driven flows, HENRIK BRUUS, KARE HARTVIG JENSEN, Department of Micro- and Nanotechnology, Technical University of Denmark, TOMAS BOHR, Department of Physics, Technical University of Denmark — It has long been recognized, that the osmotic transport characteristics of membranes may be strongly influenced by the presence of unstirred concentration boundary layers adjacent to the membrane. Previous experimental as well as theoretical works have mainly focused on the case where the solutions on both sides of the membrane remain well-mixed due to an external stirring mechanism. We investigate the effects of concentration boundary layers on the efficiency of osmotic pumping processes in the absence of external stirring i.e. when all advection is provided by the osmosis itself. This case is relevant in the study of intracellular flows, e.g. in plants. For such systems, we show that no well-defined boundary layer thickness exists and that the reduction in concentration can be visualized by a surprisingly simple mathematical relation across a wide range of geometries and Peclet numbers. This work is accepted for publication in Journal of Fluid Mechanics.
9:05AM MM.00006 Micro rheology of soft glassy materials: Collective phenomena in concentrated emulsions. NORA BENNANI, LOF, unité mixte Rhodia-CNRS-Bordeaux 1, 178 avenue du Docteur Schweitzer, F-33608 Pessac cedex - France, ANNIE COLIN, LOF — In previous studies, non local effects have been evidenced and modeled [1, 2]. The fluidity is the number of rearrangement of droplets by unit of time, and appears to be a critical parameter to describe experimental data. In the present work, flows of flocculated and non-flocculated concentrated emulsions within Hele-Shaw cells are investigated. Fast confocal imaging is used to determine fluidity and cooperative length scales, and the kinetic Elasto-Plastic model is challenged for systems with different interdroplet attractions


9:18AM MM.00007 Topology Optimization of Solid Oxide Fuel Cells¹, GRIGORIOS PANAGAKOS, FRIDOLIN OKKELS, DTU Nanotech, Department of Micro- and Nanotechnology, Technical University of Denmark, Denmark — We present a free form optimization of Solid Oxide Fuel Cell models, using a high-level implementation of topology optimization according to [1]. As a first step towards the cell’s full optimization, we focus in the design of the interconnect. The interconnect is a key element of the whole cell as it is responsible for separating the anode of one cell from the cathode of its next one in a stack of cells, thus being responsible for the supply of fuel and cooling gases, securing in the same time the efficient conductance of electrons through the stack. Modeling the steady-state operation of a fuel cell incorporates the coupling between different physics, such as reaction, electronic, ionic, thermal and fluid phenomena, and is adequately described in two dimensions. Different objective functions, guiding the optimization method, are being investigated, such like the cell’s and interconnect’s efficiency, heat convection rate and the inlet flowrates of fuel and cooling gases.


9:31AM MM.00008 Traffic jams and intermittent flows in microfluidic networks. NICOLAS CHAMPGANE, ROMAIN VASSEUR, ADRIEN MONTOURCY, DENIS BARTOLO, ESPCI ParisTech — We investigate both experimentally and theoretically the traffic of particles flowing in microfluidic obstacle networks. We show that the traffic dynamics is a non-linear process: the particle current does not scale with the particle density even in the dilute limit where no particle collision occurs. We demonstrate that this non-linear behavior stems from long range hydrodynamic interactions. Importantly, we also establish that there exists a maximal current above which no stationary particle flow can be sustained. For higher current values, intermittent traffic jams form thereby inducing the ejection of the particles from the initial path and the subsequent invasion of the network. Eventually, we put our findings in the broader context of the transport processes of driven particles in low dimension.

9:44AM MM.00009 Efficiency of a minimal micro-pump in a viscoelastic fluid, FILIPPO DE LILLO, GUIDO BOFFETTA, Dipartimento di Fisica Generale, Università di Torino, and INFN — We perform a numerical simulation of a simple micro-pump, and compare its performance in a newtonian and in a viscoelastic fluid. The device consists of two beads which are moved by means of elastic forces. While one bead is held by a static potential, the second one is moved by a pair of potentials alternating periodically. Such system was shown in [1] (where it was named the “dimer”) to be capable of forcing a newtonian fluid. We investigate the behaviour of the dimer in a viscoelastic fluid, studying how the rheology of the fluid affects the efficiency of pumping.


9:57AM MM.00010 Compliant synthetic cilia induce deposition of solid particles¹, JACLYN BRANSCOMB, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using computational modeling, we examine flow of neutrally buoyant micrometer-sized particles in a fluid-filled microchannel lined with regularly-spaced compliant cilia. The flow is driven by a pressure gradient along the channel. Our simulations reveal that non-motile synthetic cilia can be harnessed to regulate deposition of solid particles. We show that elastic cilia, deflected by the flow, create circulatory secondary flows that direct solid particles towards the ciliated channel wall, thereby inducing their rapid deposition. Our results suggest that synthetic ciliated surfaces could be harnessed for hydraulic separation, trapping, and filtration of microscopical biological and synthetic particles in microfluidic devices. The results are also useful for the understanding of the function of certain suspension-feeders that use ciliary filters to capture food particles from streaming water.

¹Supported by the Danish Agency for Science, Technology and Innovation.

Tuesday, November 23, 2010 8:00AM - 10:10AM —
Session MN Quantized Vortices and Vortex Dynamics Long Beach Convention Center 202C

8:00AM MN.00001 Quantum length scale to distinguish between Kolmogorov and Vinen forms of quantum turbulence. KATEPALLI SREENIVASAN, New York University, LADISLAV SKRBEK, Charles University, Prague — Quantum turbulence (QT) is the motion of a tangle of thin vortex filaments spontaneously generated in superfluids. Once created, the self-interaction of vortex lines in the tangle will itself create conditions for the decay of its energy. Two robust asymptotic decay laws appear in past experiments, with the vortex line density decaying either as the -1 or the -3/2 power of time. We define a new quantum length-scale which demarcates these two types of decaying QT, denoted as the tangle will itself create conditions for the decay of its energy. Two robust asymptotic decay laws appear in past experiments, with the vortex line density decaying either as the -1 or the -3/2 power of time. We define a new quantum length-scale which demarcates these two types of decaying QT, denoted as the
devices. The results are also useful for the understanding of the function of certain suspension-feeders that use ciliary filters to capture food particles from streaming water.

8:13AM MN.00002 Particles for flow visualization and velocimetry in liquid nitrogen. ENRICO FONDA, Università di Trieste - University of Maryland, College Park, MATTHEW S. PAOLETTI, University of Maryland, College Park, KATEPALLI R. SREENIVASAN, New York University, DANIEL P. LATHROP, University of Maryland, College Park — Liquid nitrogen may be used to generate, in a facility of a given size, Reynolds numbers which are substantially larger than that in water because its kinematic viscosity is one fifth that of water at 25 °C. We present a simple technique, previously used in liquid helium [1,2], to create solid tracers for visualization and velocimetry in turbulent liquid nitrogen. These tracers are created by injecting a gaseous mixture of room-temperature nitrogen and an additional gas (element or compound) into the flow. The latter is selected in order to create circulatory secondary flows that direct solid particles towards the ciliated channel wall, thereby inducing their rapid deposition. Our results suggest that synthetic ciliated surfaces could be harnessed for hydraulic separation, trapping, and filtration of microscopical biological and synthetic particles in microfluidic devices. The results are also useful for the understanding of the function of certain suspension-feeders that use ciliary filters to capture food particles from streaming water.

8:26AM MN.00003 Reconnection of quantized vortices and quantum turbulence1, DANIEL LATHROP, University of Maryland — Turbulence in superfluid $^4$He is dominated by reconnection and ring collapse. We utilize micron and nano-scale ice particles to visualize the dynamics of quantized vortices and the normal component. After briefly reviewing our observations of these phenomena, I will discuss reconnection dynamics at large and small scales. Those dynamics can be understood using scaling solutions and some ideas from dynamical systems. There is one underlying question we work to address: is there a single universal reconnection dynamics, do we need to consider a one or two parameter family of reconnection events?

1This work was done in collaboration with M.S. Paolelli, M.E. Fisher, and K.R. Sreenivasan, and is supported by the NSF DMR.

8:39AM MN.00004 Visualization of the Quantized Vortex Lattice Dynamics in $^3$He, KRISTINA GAFF, DANIEL LATHROP, University of Maryland, College Park — We study the lattice structure and dynamics of quantized vortices in superfluid helium using a new rotating experiment. This setup includes control of the entire apparatus from the rotating frame as well as implementation of a novel isolation cell, which permits investigation into new phenomena such as differential rotation in helium-II. Our documentation of the vortex lattice dynamics in the $(r, \phi)$ plane (i.e. longitudinal to the vortices) includes real-time visualization of Tkachenko waves as well as evidence of differential rotation with distinct Stewartson layer boundaries. We also present possible Kelvin-Helmholtz instabilities and the formation and propagation of superfluid vortex bundles. We show that the angular velocity is a function of radius and may be driven by the geometry of the isolation cell.

8:52AM MN.00005 Initial conditions for reconnection calculations of quantized vortices, CECILIA RORAI, Università degli Studi di Trieste - University of Maryland, DANIEL P. LATHROP, MICHAEL E. FISHER, University of Maryland, KATEPALLI R. SREENIVASAN, New York University — Vortex reconnection occurs when two vortices intersect and then rejoin with exchanged tails. This process occurs in both classical fluids and superfluids, in superconductors, and in magnetized plasmas. In helium II this phenomenon has been investigated numerically via the Gross-Pitaevskii equation. Although a simplified model, the Gross-Pitaevskii equation is interesting since it naturally embodies vortex reconnection, as first numerically shown by Koplik & Levine [Phys. Rev. Lett. 71, 1375 (1993)]. A crucial issue in such computations is the selection of initial conditions. The question we address is how initial wave functions may effect the outcome of reconnection. Traditionally the initial conditions have been generated by multiplying approximate wave functions for a single vortex and imposing, to some degree, periodic boundary conditions. An alternative approach will be presented. It consists in selecting an initial configuration that minimizes the total energy. The differences between the results obtained with this approach and previous ones will be discussed especially with respect to the dispersion of energy seen in quantum turbulence.

9:05AM MN.00006 Motion of a Vortex Filament in the Local Induction Approximation: Reformulation of the Da Rios-Betchov Equations in the Extrinsic Filament Coordinate Space, BHIMSEN SHIVAMOGGI, University of Central Florida, GERT JAN VAN HEIJST, Eindhoven University of Technology, Eindhoven, The Netherlands — In recognition of the highly non-trivial task of computation of the inverse Hasimoto transformation mapping the intrinsic geometric parameter space onto the extrinsic vortex filament coordinate space a reformulation of the Da Rios-Betchov equations in the latter space is given (Shivamoggi and van Heijst [1]). The nonlinear localized vortex filament structure solution given by the present formulation is in detailed agreement with the Betchov-Hasimoto solution in the small- amplitude limit and is shown in qualitative agreement with laboratory experiment observations of helical-twist solitary waves propagating on concentrated vortices in rotating fluids. The present formulation also provides for a discernible effect of the slipping motion of a vortex filament on the vortex evolution via an amplitude change in the vortex kink.


9:18AM MN.00007 On Lagrangian and vortex-surface fields in Taylor-Green and Kida-Pelz flows1, DALE PULLIN, YUE YANG, California Institute of Technology — A methodology is developed for constructing smooth scalar fields $\phi$ for Taylor-Green and Kida-Pelz velocity fields that, at $t = 0$, satisfy $\omega \cdot \nabla \phi = 0$. We refer to such fields as vortex-surface fields. Iso-surfaces of $\phi$ then define vortex surfaces. Given the vorticity, our definition of a vortex-surface field is shown to admit nonuniqueness, and this is resolved numerically using an optimization approach. Equations describing the evolution of vortex-surface fields are obtained for both inviscid and viscous incompressible flows. For the former, the Helmholtz vorticity theorem shows that Lagrangian material surfaces which are vortex (or vorticity) surfaces at the initial time remain so for later times. By tracking $\phi$ as a Lagrangian field in slightly viscous flows, we show that the well-defined evolution of Lagrangian surfaces that are initially vortex surfaces can be a good approximation to vortex surfaces at later times prior to vortex reconnection. In the evolution of such Lagrangian fields, we observe that initially blob-like vortex surfaces are progressively stretched to sheet-like shapes, with subsequent rolling up of structures near the interface. The non-local geometry in the evolution is quantified by differential geometry properties.

1Supported by the NSF.

9:31AM MN.00008 Quasi-steady linked vortices with chaotic streamlines, OSCAR VELASCO FUENTES, ANGELICA ROMERO ARTEAGA, CÍCÉSE — We study the dynamics of two or more toroidal filamentary vortices — i.e. thin tubular vortices coiled on an immaterial torus— in an otherwise quiescent, ideal fluid. Assuming that the vortices are identical and equally spaced on a meridional section of the torus, the flow evolution depends on the torus aspect ratio ($R$), the number of vortices ($N$), and the vortex topology ($p,q$), where $p$ and $q$ are coprime integers such that the $V_{p,q}$ vortex winds $p$ times around the torus symmetry axis and $q$ times around the torus centerline). The evolution of sets of $V_{1,1}$ and $V_{1,2}$ vortices was computed using the Rosenhead–Moore approximation to evaluate the velocity field and a four-fold order Runge-Kutta scheme to advance in time. It was found that vortex sets with $N > 6$ and $R < 0.15$ progressed along and rotated around the torus symmetry axis in an almost steady manner while each vortex in the set appropriated preserved its shape. The velocity field, observed in the comoving frame, has two stagnation points. The stream tube starting at the forward stagnation point and the stream tube ending at the backward stagnation point transversely intersect along a finite number of streamlines. The three-dimensional chaotic tangle that arises has a geometry which depends primarily on the number of vortices $N$.

9:44AM MN.00009 Dynamics of Oscillatory Vortex Multipoles Generated by Electromagnetic Forcing, ALDO FIGUEROA, SERGIO CUEVAS, EDUARDO RAMOS, CIE-UNAM — Vortices formed by the concurrent effect on a localized magnetic field distribution and two alternate electric currents perpendicular to each other in a shallow (4mm) layer of an electrolyte are analyzed. Alternate currents with frequencies and amplitude in the range of 1-500 mHz and 80 mA, respectively, are explored. For a single dipolar magnetic field and a single electric current, the dominant structure of the flow is a pair of alternating lobes located co-linear with the generated Lorenz force. The flow presents a resonant behavior when the forcing frequency is around 10 mHz. When multipoles are used to generate the magnetic field, more complicated lobe distributions are obtained. The flow patterns were successfully described using a quasi-two-dimensional numerical model. A tridimensional numerical models corroborates the theoretical results. Flow visualization and numerical Lagrangian particle tracking indicate that multipolar flows present symmetries according to the magnetic field distributions. Although in some regions the flow patterns efficiently mix the fluid, the mixing is inhomogeneous due to symmetry conditions of the flows. Mixing is enhanced when symmetries are destroyed by the use of a random array of magnets or by injecting two electric currents.
9:57AM MN.00010 Vortical structures in the shallow flow past a magnetic obstacle in an electrolytic layer, ALBERTO BELTRAN, Universidad Nacional Autonoma de Mexico and University of California, Los Angeles, SERGIO CUEVAS, EDUARDO RAMOS, Universidad Nacional Autonoma de Mexico, SERGEY SMOLENTSEV, University of California, Los Angeles — It is known that the interaction of electric currents (induced or injected) and a localized magnetic field produces a Lorentz force that inhibits the motion of the fluid and acts as an obstacle for the flow (a magnetic obstacle). In this work, the flow in a shallow electrolytic layer produced by a uniform injected current and a localized non-uniform magnetic field is simulated numerically using quasi-twowimensional and three-dimensional models, with a parallelized version of the numerical code. Different vortex patterns that have been observed experimentally in the wake of the magnetic obstacle are obtained, including steady vortex dipoles and vortex shedding flow. The three dimensional structure of the flow is explored and, particularly, the velocity profiles in the layer depth showing the appearance of inflection points that determine the stability properties of the flow, are analyzed.

Tuesday, November 23, 2010 8:00AM - 10:10AM — Session MP Microfluids: Mixing I Long Beach Convention Center 203A

8:00AM MP.00001 Breakup of particle clumps on liquid surfaces, SATISH GURUPATHAM, M.M. HOSSAIN, B. DALAL, I. FISCHER, P. SINGH, NITT, D.D. JOSEPH, University of Minnesota — Although it is known that a clump of powder floating on a liquid surface breaks up to form a monolayer of particles on the surface, the mechanism by which this happens is not entirely understood. We show that when a floating clump comes in contact with the liquid surface particles on its outer periphery are pulled into the interface by the capillary force overcoming the cohesive forces that keep the clump together. Furthermore, the newly adsorbed particles move away from the clump which is a consequence of the fact that when a particle is adsorbed on to a liquid surface it causes a flow away from itself on the interface. This flow causes the newly adsorbed particles to not only move away from each other, but also away from the clump. Interestingly, when many particles are asymmetrically broken apart from the clump, the clump itself is moved away by the flow due to the newly adsorbed particles.

8:13AM MP.00002 Electro-osmotic flows in rectangular cavities, VIATCHESLAV MELESHKO, Dept Theoretical and Applied Mechanics Kiev National University, ALEXANDRE TROFIMCHUK, Dept Natural Resources Institute of Telecommunication and Global Information Space NAS Ukraine, ALEXANDRE GOURJII, Dept Vortex Motion Institute of Hydromechnanisication and Global Information Space NAS Ukraine, ELINA BEZYM’YANA, Dept Theoretical and Applied Mechanics Kiev National University — The talk presents the results of investigation of the microfluidics mixing processes in a rectangular cavity flows induced by electro-osmotic excitation. Enhanced mixing plays an important role in biological and chemical pharmaceuticals analysis in microfluidics systems. Analytical solution is presented for the velocity field in the cavity under various electric potential distributions. The location of the periodic points in the flow are accurately established and the structure of stable and unstable manifolds is discussed. The optimal form of excitation is suggested in order to obtain most effective mixing regime in the cavity. The regular and chaotic regions are identified under various condition of excitation. Finally, we compare numerical and analytical solutions with the results of laboratory experiments for real microfluidic flows.

8:26AM MP.00003 Targeting Complete Chaotic Mixing by Destabilizing Key Periodic Orbits in an Electro-osmotic Mixer, RODOLPHE CHABREYRIE, Carnegie Mellon University, Mechanical Engineering Department, Pittsburgh, PA 15213, CRISTEL CHANDRE, Centre de Physique Théorique, CNRS-Aix-Marseille Université, Campus de Luminy, case 907, F-13288 Marseille cedex 09, FRANCE, PUSHPENDRA SINGH, New Jersey Institute of Technology, Mechanical Engineering Department, NADINE AUBRY, Carnegie Mellon University, Mechanical Engineering Department, Pittsburgh, PA 15213 — The ability to generate complete, or at least well spread, chaotic mixing is of great interest in numerous applications, especially microfluidics. For this purpose, we propose a strategy that allows us to quickly target the parameter values at which complete mixing occurs. The technique is applied to a time periodic, two-dimensional electro-osmotic flow with spatially and temporally varying Helmholtz-Smoluchowski slip conditions. The strategy consists of following the linear stability of some key periodic orbits in parameter space, particularly identifying bifurcation points at which such orbits become unstable. Poincaré maps, Lyapunov exponents and a box counting measure are all computed to validate the strategy.

8:39AM MP.00004 Quantitative description of mixing in non-perturbative flows, RADFORD MITCHELL, ROMAN GRIGORIEV, Georgia Tech — Studies of near-integrable fluid flows have identified two independent and complementary quantitative metrics which are needed to describe their mixing properties: (1) the mixing rate and (2) the size (and shape) of the mixed region. In the limit of weak perturbation away from integrability both metrics can be computed using multi-scale averaging theory. In this talk we show how these metrics can be computed in the non-perturbative regime using the formalism of Periodic Orbit Theory. Using a Taylor-Couette-like steady flow as an example, we show that the boundaries of the mixed region are formed by heteroclinic manifolds of periodic (or relative periodic) orbits while the mixing rate can be quantified using a weighted sum over a set of both periodic and relative periodic orbits. A similar description is expected to be valid for time-periodic flows and other geometries as well.

8:52AM MP.00005 Characterization on performance of micromixer using DC-biased AC electroosmosis, BI-Q PARK, SIMON SONG, Hanyang University — An active micromixer using DC-biased AC-Electroosmosis (ACEO) is investigated to figure out the effects of design parameters on the mixing performance. The mixer consists of a straight microchannel, with a cross section of 60 x 100 μm, and gold electrode pairs fabricated in the microchannel. The design parameters include the number of electrode pairs, flow rate, DC-biased voltage, AC voltage and AC frequency. First, we found that a mixing index became 80% 100 μm downstream of a single electrode pair with a length of 2 mm when applying a 25Vpp, 2.0 VDC, 100 kHz sine signal to the electrodes. With decreasing AC frequency, the mixing index was affected little. But the mixing index significantly increases with increasing either DC-biased voltage or AC voltage. Also, we were able to increase the mixing index up to 90% by introducing alternating vortices with multiple electrode pairs. Finally, we discovered that the mixing index decreases as the flow rate increases in the microchannel, and there is an optimal number of electrode pairs with respect to a flow rate. Detailed quantitative measurement results will be presented at the meeting.

9:05AM MP.00006 Coherent structures in 3D viscous time-periodic flow, J.G. ZNAJEN, M.F.M. SPEETJENS, R.R. TRELING, H.J.H. CLERCX, Dept. of Physics, Eindhoven University of Technology, P.O.Box 513, 5600MB Eindhoven, The Netherlands — Periodically driven laminar flows occur in many industrial processes from food-mixing devices to micro-mixer in lab-on-a-chip systems. The present study is motivated by better understanding fundamental transport phenomena in three-dimensional viscous time-periodic flows. Both numerical simulation and three-dimensional Particle Tracking Velocimetry measurements are performed to investigate the 3D advection of a passive scalar in a lid-driven cylindrical cavity flow. The flow is forced by a time-periodic in-plane motion of one endwall via a given forcing protocol. We concentrate on the formation and interaction of coherent structures due to fluid inertia, which play an important role in 3D mixing by geometrically determining the tracer transport. The disintegration of these structures by fluid inertia reflects an essentially 3D route to chaos. Data from tracking experiments of small particles will be compared with predictions from numerical simulations on transport of passive tracers.
9:18 AM MP.00007 High-throughput continuous millisecond solution exchange for particle suspensions. DANIEL GOSSETT, HENRY TSE, JAIDEEP DUDANI, DINO DI CARLO, UCLA — Mixing and solution exchange are routine tasks in macro-scale systems. However, they can be challenging and slow in microfluidic systems which lack turbulence, and mixing is often achieved by manipulation of passive diffusion. In previous work we characterized geometry-dependent inertial lift forces which act on objects entrained in high velocity flows in confined microchannels. Using inertial focusing we can precisely focus spherical particles or cells to equilibrium positions with throughputs of thousands per second. Inertial effects have been employed in microfluidic systems for membrane-free filtration, size-based sorting, and cytometry. Here, we present a novel microfluidic system which manipulates these lift forces to rapidly transfer particles and cells from solution to reporter by geometrically defining a new equilibrium position in a cold flow in curved channels. This behavior requires fluid inertia and micro-particles offset from the channel center, spinning due to the local shear rate across the particle. This phenomenon provides a novel technique for fluid transfer and solution switching in microsystems. Compared to other microfluidic mixing approaches the technique requires a simple channel geometry, with no external forces for operation, enabling biological applications in which the cells present within the flow themselves can induce the fluid transfer required for labeling, lysis, and other high-throughput sample preparations.

9:31 AM MP.00008 Spinning Convection: Particle-induced Secondary Flow in Straight Microchannels. HAMED AMINI, ELODIE SOLLIER, DINO DI CARLO, University of California, Los Angeles — In microfluidic systems, flow is generally approximated with Stokes flow and inertial forces are assumed negligible. However, at finite Reynolds number (small, yet non-zero), inertial forces have been shown to be useful, for instance by causing randomly distributed particles to order on specific lateral equilibrium positions in a confined flow. To further study these inertial effects at the microscale, we investigated the local disturbances induced by spinning particles in straight microchannels. We observe, numerically and experimentally, unexpected net cross-stream disturbances that generate a unique secondary flow pattern, which resembles the well-known Dean flow in curved channels. This behavior requires fluid inertia and micro-particles offset from the channel center, spinning due to the local shear rate across the particle. This phenomenon provides a novel technique for fluid transfer and solution switching in microsystems. Compared to other microfluidic mixing approaches this technique requires a simple channel geometry, with no external forces for operation, enabling biological applications in which the cells present within the flow themselves can induce the fluid transfer required for labeling, lysis, and other high-throughput sample preparations.

9:44 AM MP.00009 Inertial Droplet Mixing in a Confined Microchannel Gas Flow. BRIAN CARROLL, BRIAN ROBINSON, CARLOS HIDROVO, The University of Texas at Austin — Efficient mixing at the microscale remains a formidable engineering challenge. Recent advancements in the field of microfluidics have been focused on inertial effects. Inertial effects can be used to enhance mixing in microchannels, particularly for high Peclet number, convection dominated mixers. In this work, we present results of a novel droplet-based mixing technique currently being developed, which allows for high mixing rates and low energy consumption. We consider the flow created by two parallel streams where the Reynolds number is so low that the natural flow instabilities are completely damped and mixing occurs only at the interface between the two streams. Our interest is in increasing the number of mixing interfaces, in order to enhance the amount of molecular exchange, by imposing an external perturbation on the flow streams. The flow visualization is used to investigate the general features of the flow, while chemically reacting LIF is employed to quantify the extent of molecular mixing. Results will be presented over a range of perturbation frequencies and amplitudes.

9:57 AM MP.00010 Mixing Enhancement in a Very Low Reynolds Number Liquid Shear Flow. ROHIT NEHE, Michigan State University, HUI HU, Iowa State University, MANOOCHEHR KOOCHEFSAHANI, Michigan State University — We consider the shear flow created by two parallel streams where the Reynolds number is so low that the natural flow instabilities are completely damped and mixing occurs only at the interface between the two streams. Our interest is in increasing the number of mixing interfaces, in order to enhance the amount of molecular exchange, by imposing an external perturbation on the flow streams. The flow visualization is used to investigate the general features of the flow, while chemically reacting LIF is employed to quantify the extent of molecular mixing. Results will be presented over a range of perturbation frequencies and amplitudes.

8:00 AM MQ.00001 Low Reynolds number swimming in a stratified fluid. AREZOO ARDEKANI, ROMAN STOCKER, Massachusetts Institute of Technology — Microorganisms live in aquatic environments that are often density-stratified, for example due to temperature or salinity gradients in oceans and lakes. Yet, the effect of stratification on low-Reynolds number swimming has not been investigated, in part because it is generally believed that the length scale of stratification is orders of magnitude larger than microorganisms. We show that this is incorrect and that typical stratifications can affect organisms as small as O(100 µm). By deriving fundamental singularity solutions (Stokeslet and stresslet) in a stratified fluid—which we call Stratlets—we demonstrate that the characteristic length scale of this problem, \( L = (\mu \gamma / h)^{1/4} \), is one that combines buoyancy, diffusion and viscosity effects, where \( \alpha \) is the diffusivity of the stratifying agent, \( \mu \) the dynamic viscosity, \( \gamma \) the background density gradient, and \( g \) the acceleration of gravity. The importance of stratification for a swimmer of size \( a \), relative to diffusion, is measured by the Rayleigh number, \( Ra = (a/L)^{4} \). Stratification dramatically changes the flow generated by a swimmer, creating recirculation cells that diminish in size with increasing \( Ra \). Consequently, flow velocity decays with distance from the swimmer considerably faster than in homogeneous fluids. This suggests that stratification acts as a “silencer” for hydromechanical signals, for example reducing the perception abilities of microorganisms that rely on mechanosensing to detect prey.

8:13 AM MQ.00002 Locomotion by tangential deformation in a polymeric fluid. LAILAI ZHU, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden, ERIC LAUGA, University of California San Diego, LUCA BRANDT, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden — Many biological cells such as bacteria often encounter viscous environments with high stresses. The physics of micro-propulsion in such a non-Newtonian viscoelastic fluid has only recently started to be addressed. Here we present results of three-dimensional numerical simulations for the steady locomotion of a self-propelled body in a model polymeric (Giesekus) fluid at low Reynolds number. The microswimmer is driven by a purely tangential distortion on the outer surface produced as non-homogenous boundary condition on a rigid body. The swimming speed and efficiency for different values of the Weissenberg number and the viscosity ratio are reported. The swimming speed is lower in a visco-elastic fluid and is asymptotically recovering for large Wi approaching values for Newtonian swimmer. Interestingly, the efficiency is seen to significantly increase as the viscosity of the polymeric fluid is increased. Further analysis reveals that polymeric stresses break the Newtonian front-back symmetry in the flow profile around the body. Speed and efficiency for pusher and puller swimmers will be reported together with analysis of the velocity fields. Time-dependent boundary conditions shall also be considered.

8:26 AM MQ.00003 Undulatory Swimming in Viscoelastic Fluids at Low Reynolds Number. XIAONING SHEN, PAULO ARRATIA, University of Pennsylvania — In this talk, we present an experimental investigation on the swimming behavior of the worm nematode C. elegans in viscoelastic fluids at low Reynolds numbers. The C. elegans swimming behavior is characterized by tracking the nematode body posteriors and using particle image velocimetry. Results show that the nematode responds to the fluid elastic stresses by adjusting its beating frequency and wave-form. Overall, low levels of elasticity tend to hinder swimming speed by 30% when compared to a Newtonian fluid with similar viscosity. These results, however, are only valid for Wissenberg numbers below unit (\( Wi<1 \)), where \( Wi \) is defined as the product of the fluid relaxation time and the fluid flow shear-rate.

1 NSF CAREER CBET 0954084

8:35 AM MP.00011 Millisecond Solution Exchange for Particle Suspension. HONG ZHAO, ROHIT NEHE, Michigan State University, HUI HU, Iowa State University, MANOOCHEHR KOOCHEFSAHANI, Michigan State University — The high Peclet number convection is dominant and diffusion is negligible for the length of the channel. We intend to use this technique for on-chip, in-line sample preparation or for studying reaction kinetics and molecular binding events where rapid and complete solution exchange or mixing is required.
shear flow are crucial to understand cell attachment at the onset of biofilm formation. We combined microfluidics and holography to measure 3-D trajectories of bacteria and track swimming behavior and escape distances of single swimming cells of the bacterium *Escherichia coli*. By using near-infrared lasers and a two-photon excitation microscopy system, we were able to perform real-time 3-D tracking of swimming trajectories near the wall of a microfluidic channel. Our studies showed that bacteria can escape from shear flows at increased cell speeds, and that swimming speed and escape distance are conserved in this process. Interestingly, we observed no difference in the number of swimming strokes per escape event. Instead, the animals exhibit a compensatory mechanism near the wall to maintain swimming distance.

---

**9:05AM MQ.00006** Marine oyster swimming behavior in the benthic boundary layer under different flow conditions, KELLY SUTHERLAND, JOHN DABIRI, California Institute of Technology, MIMI KOEHL, University of California, Berkeley — Marine organisms swimming in water near the substratum are subjected to boundary layer flow, which is characterized by steep velocity gradients and turbulence. How do small swimming organisms navigate flows at this interface and interact with mates? We recorded in the field the swimming behavior of marine oysters near complex living substrata exposed to different ambient water flow conditions. Oyster trajectories and background water flow were recorded simultaneously using a Self-Contained Underwater Velocimetry Apparatus (SCUVA). Particle image velocimetry enabled us to map the instantaneous water velocity fields in which the oysters were swimming. In slow flows ($U_{rms} \sim 0.3$ cm s$^{-1}$), oyster swimming tracks were more tortuous, and encounters with bottom-dwelling organisms and other oysters were more frequent than in higher velocity wave-driven flows ($U_{rms} \sim 2.8$ cm s$^{-1}$), indicating that foraging and mating activities may be curtailed when ambient water flow is too rapid or variable.

---

**9:18AM MQ.00007** Escaping From Predation At Low Reynolds Number: A Compensatory Mechanism, BRAD GEMMELL, University of Texas at Austin, JIAN SHENG, University of Minnesota, ED BUSKEY, University of Texas at Austin — Small planktonic organisms such as copepods are often the first foods for many species of fish and thus, subject to high predation rates. They have developed strong escape responses to attacks from visual predators and this behavior is found even in the youngest development stage. Because of their small size (approx. 100 µm), these juvenile copepods must contend with greater viscous forces than their predators during encounters. In this study, we investigate the role of viscosity on escape swimming performance of young copepods within the context of the environmental temperatures (10C-30C) these animals experience along the Texas coast. 3-Dimensional high speed (3000 frames per second) digital holographic techniques were used to elucidate kinematics and kinetics of swimming. Here we show that although escape velocity and acceleration are reduced as a function of both increasing viscosity and decreasing temperature, total escape distance is conserved. Interestingly, we observed no difference in the number swimming strokes per escape. Instead, the animals exhibit a compensatory mechanism based on increasing power stroke duration to recovery stroke duration to counteract the increasing viscosity at lower temperature. Flow analysis shows this results in the conservation of energy expenditure, and consequently escape distance.

---

**9:31AM MQ.00008** A High-Speed Tomographic PIV System for Measuring Plankton-Generated Flow, D.W. MURPHY, D.R. WEBSTER, J. YEN, Georgia Tech — Plankton such as copepods, fish larvae, and mysids occupy a fluid environment in which neither inertia nor viscosity dominates. At this intermediate Reynolds number (range of 1 to 1000), locomotion, hydrodynamic signal detection, and foraging of these organisms are influenced by both viscous and inertial effects. The millimeter length and millisecond time scales at which these animals operate present significant difficulties to both flow measurements using traditional planar PIV systems, which additionally cannot quantify the three-dimensional nature of the flow. We describe the design and application of a novel PIV system comprising four high-speed cameras (2190 fps), two near-IR lasers (808 nm), and the associated optics used to illuminate and interrogate a volume of approximately 1 cubic centimeter. Illumination in the near IR wavelengths does not affect copepod behavior. Fine-scale three-dimensional fluid velocity measurements around free-swimming animals provide insight into their locomotion-induced flow. Further, calculation of the complete strain rate tensor and vorticity vector allows estimation of the flow disturbance and mechanosensory reaction levels. The system also facilitates studies of organism response to environmental cues such as laboratory-generated turbulence.

---

**9:44AM MQ.00009** Bacteria swimming in a wall-bounded shear flow studied with microfluidic-DHM1, H. AGARWAL, U. of Minnesota, M. BARRY, R. STOCKER, MIT, J. SHENG, U. of Minnesota — Observations of bacterial motility in a wall-bounded shear flow are crucial to understand cell attachment at the onset of biofilm formation. We combined microfluidics and holography to measure 3-D trajectories of *Escherichia coli* in shear flows, for shear rates up to 200/s. Acquisition of >3,000 trajectories over short times (5 min) enabled the robust quantification of swimming velocities and dispersion coefficients. We find that swimming, including swimming in circles and other trajectories, is driven by viscous forces from the wall. The flow direction affects the swimming direction, consistent with previous studies.

---

**9:57AM MQ.00010** Autonomous motion of semipermeable colloidal particles via chemical reactions: self-osmophoresis, MISAEL DIAZ, UBALDO CORDOVA-FIGUEROA — While a large body of work exists on the design of catalytically-driven colloidal particles, little work exists on particles with the ability to permeate fluid through its surface that may be used for applications in lab-on-a-chip systems and drug delivery. We propose a model for the catalytically-driven motion of a semipermeable particle (e.g., non-motile microorganisms and vesicles) surrounded by reactant solutes in a Newtonian fluid. It is assumed that a first-order consumption reaction of surrounding reactants—which could be enzymatic or catalytic—occurs on half of the outer surface of the particle. In equilibrium, the osmotic pressure inside the particle balances that of outside. The reaction creates an imbalance in osmotic pressure, causing fluid to permeate the side of the particle to balance the internal fluid. This fluid motion satisfies mass conservation inside the particle, causing particle motion towards regions of low reactant concentration by a mechanism known as osmophoresis. Preliminary results show that the particle velocity—defined as a Péclet number—is a function of the permeability of the membrane, a characteristic osmotic velocity, and the Damköhler number—which is a measure of relative impacts of the diffusion and chemical reaction. The permeating fluid retards particle motion by dragging the solute against the induced osmotic imbalance.

---

1 Supported by NIH and NSF
8:00AM MR.00001 DNS of high speed boundary layers over ablating surfaces, KALEN BRAMAN, VENKAT RAMAN, Department of Aerospace Engineering and Engineering Mechanics, The University of Texas at Austin, ROCHAN UPADHYAY, PECOS Center, The University of Texas at Austin, OFODIKE EZEKOYE, Department of Mechanical Engineering, The University of Texas at Austin — Ablation of thermal protection shields is an important design problem in developing reentry vehicles. Development of predictive computational models for this problem will enable optimization of the size and hence weight of the protective layer. In this work, direct numerical simulation (DNS) of a compressible ablating boundary layer is used to understand the modeling issues in the context of Reynolds-averaged Navier Stokes (RANS) equations. The DNS is performed at conditions obtained from a detailed RANS study of a reentry vehicle. The free stream conditions of the two simulations are Mach 0.6, temperature 5940 K, and $Re_{T}=1000$; and Mach 1.2, temperature 5580 K, and $Re_{T}=2000$. The surface ablation of a graphite ablator is modeled using a locally 1-D, quasi-steady state formulation with control volume mass and energy balances over the interior of the ablator. A 10-species gas phase chemistry mechanism is used. A priori studies are used to evaluate scalar flux models and the reaction source term closure in RANS.

8:13AM MR.00002 Direct Numerical Simulation of low-temperature ablation by turbulent flows1, RYAN CROCKER, YVES DUBIEF, University of Vermont, CHRISTOPHER WHITE, University of New Hampshire — The present study is motivated by the understanding and modeling of the dynamic interactions between a turbulent fluid transporting an erosive agent, and an erodible surface. As the erosive agent causes changes in the geometry of the wall-boundary conditions, turbulence may rapidly evolve into a non-equilibrium state and may further accelerate ablation. To investigate this complex process, a direct numerical simulation (DNS) algorithm is designed to simulate the temporal and spatial evolution of a surface subjected to low-temperature ablation caused by turbulent flow. The ablative wall is fully discretized and the interface fluid/wall is modeled by a level-set method combined with flow and thermal immersed boundary methods. After a discussion of numerical challenges and their solutions, low Reynolds turbulent ablation flows are used to illustrate the complexity of the problem with a focus on emerging turbulent and topographical scales as ablation proceeds.

8:26AM MR.00003 Transport of Ablation Products in a Mach 5 Boundary Layer using Naphthalene PLIF, CLEMENS NOEL, BRYAN LOCHMAN, ZACH MURPHREE, VENKAT NARAYANASWAMY, The University of Texas at Austin, Austin, TX — Planar laser-induced fluorescence of sublimated naphthalene was used to visualize the transport of ablation products in a Mach 5 turbulent boundary layer. The naphthalene was seeded into a rectangular insert that was mounted flush with the floor of a Mach 5 wind tunnel. The naphthalene fluorescence was excited using a frequency quadrupled Nd:YAG laser (266 nm) and the fluorescence emission between 310 – 350 nm was collected. The temperature and pressure dependences of the fluorescent decay were studied in detail. An increase in the fluorescence was observed with increasing temperature for the temperature range tested (300 – 525K). Fluorescence-lifetime measurements were made in pure-air and nitrogen environments at 300 K over the range 3.3-1013 KPa. The results in air exhibited the expected Stern-Volmer behavior with decreasing lifetimes at increasing pressure, whereas nitrogen exhibited the opposite trend. Preliminary PLIF images of the sublimated naphthalene were acquired in the Mach 5 turbulent boundary layer. Relatively low signal-to-noise-ratio images were obtained at a stagnation temperature of 345 K, but much higher signal images were obtained at a stagnation temperature of 375 K.

8:39AM MR.00004 Stochastic approach of meteor-generated infrasounds, CHRISTOPHE MILLET, CHRISTOPHE P. HAYNES, CEA, DAM, DIF, F-91297 Arpajon, France — In recent years, numerous bolide sources have been detected by the International Monitoring System infrasound arrays. In the present study, the meteorite fall near Carancas, Peru, on September 15, 2007 has been analyzed through the simulation of the emission and non-linear propagation of shocks. Given that the meteoroid body shape and the way infrasounds are generated are generally not well known, the unknown parameters of bolides have been chosen to be random fields. By comparing our analytic and numeric results to recorded data, we aim to show that it might be entirely plausible that N-waves could have originated from the Carancas meteor scenario. It is shown that, given a random entry diameter, the uncertainty of the ground overpressure increases as the N-wave emission altitude decreases, as a result of bolide near-field effects, except for low-altitudes, where the uncertainties associated with the propagation medium may be dominant.

8:52AM MR.00005 Shock-wave surfing and the separation of meteoroid fragments in the atmosphere, STUART LAURENCE, RALF DEITERING — Studying the aerodynamic interactions between bodies travelling at highly supersonic speeds is necessary to our understanding of the separation of meteoroid fragments following atmospheric disruption. Here we show that a phenomenon referred to as “shock-wave surfing,” in which a body moves in such a way as to follow the shock wave generated by another upstream body, can lead to the accumulation of a significantly higher relative lateral velocity between fragments than would otherwise be possible. The surfing phenomenon is investigated for the canonical “shock-wave surfing,” in which a body moves in such a way as to follow the shock wave generated by another upstream body, can lead to the accumulation of a significantly higher relative lateral velocity between fragments than would otherwise be possible. The surfing phenomenon is investigated for the canonical cases of interactions between a sphere and a wedge, and between two spheres. Numerical simulations are performed and a simple theoretical model is developed to determine the forces acting on the surfing body. A phase-plane description is employed to elucidate features of the system dynamics in both cases. For the two sphere case, a strong influence of the body radius ratio on the separation process is found and a critical ratio is predicted for initially touching fragments that delineates entrainment of the smaller fragment within the larger fragment’s shock from expulsion. It is also shown that a large fraction of the variation in the separation behavior of meteoroid fragments deduced by previous authors from an analysis of terrestrial crater fields can be explained by a combination of surfing and a modest rotation rate of the parent body.

9:05AM MR.00006 Experimental Investigation of Shock Wave Surfing, N.I. PARZIALE, H.G. HORNUNG, J.E. SHEPHERD, California Institute of Technology, S.J. LAURENCE, DLR — Shock wave surfing is investigated experimentally in GALCIT’s Mach 4.0 Ludwig Tube. Shock wave surfing occurs when a secondary free-body follows the bow shock formed by a primary free-body; an example of shock wave surfing occurs during meteorite breakup. The free-bodies in the current investigation are nylon spheres. During each run in the Ludwig tube a high speed camera is used to capture a series of schlieren images; edge tracking software is used to measure the position of each sphere. Velocity and acceleration are had from processing the position data. The radius ratio and initial orientation of the two spheres are varied in the test matrix. The variation of sphere radius ratio and initial angle between the centers of gravity are shown to have a significant effect on the dynamics of the system.

9:18AM MR.00007 Shock Structure in a Supersonic Jet1, CATALINA STERN, CESAR AGUILAR, UNAM — We visualize the stationary shock structure of a 1.6mm supersonic jet flow using a shadowgraph. The form and size of the structure can be determined. Through the heterodyne detection of Rayleigh scattering by the flow, we can obtain the instant spatial Fourier Transform for a given wave vector, of the density fluctuations in the flow. The wave vector is related to the size of the fluctuations and indicates their direction of propagation. The spectral density of these fluctuations shows two of the modes described by Monin and Yaglom: The entropic mode is related to entropy fluctuations at constant pressure, the acoustic mode is related to isentropic pressure fluctuations. Besides the two predicted peaks, another one appears close to the shocks, that moves at slow speed. We present preliminary results that show that this slow speed fluctuation appears always close and parallel to the shock. Research is still in progress to understand the dynamics in these regions of the flow.

1We acknowledge support from DGAPA PAPIIT IN119509.
9:31AM MR.00008 Large eddy simulation of an underexpanded sonic jet , CATHERINE GORLE. Stanford University, MIRKO GAMBA, FRANK HAM, Stanford University. CENTER FOR TURBULENCE RESEARCH, PSAAP TEAM — Large eddy simulations (LES) for an underexpanded sonic jet in quiescent flow have been performed using the explicit spatially-filtered compressible Navier-Stokes solver Charles. The unstructured finite volume method uses a blended central-upwind scheme in smooth flow regions to minimize artificial damping of resolvable turbulence scales and switches to a third order WENO method and an HLCC approximate Riemann solver to capture discontinuities. Time discretisation is performed with an explicit third order Runge Kutta scheme. The simulations reproduce the conditions of an experiment where single-shot Schlieren imaging of the jet is used to investigate the instantaneous and time-averaged steady-state structure of the barrel shock and the jet far-field growth rate. A pseudo-time sequence of the formation of the barrel shock that tracks the jet injection transient is also constructed. A comparison of the shock structure and time scale of the jet formation obtained from the LES and the experiment is presented, showing a good agreement in the shock structure. Future work includes a similar study for an underexpanded sonic jet in a supersonic cross flow, and will also focus on the investigation of turbulent mixing.

9:44AM MR.00009 Ignition, Flame Structure and Near-Wall Burning in Transverse Hydrogen Jets in Supersonic Crossflow 1, MIRKO GAMBA, M. GODFREY MUNGAL, RONALD K. HANSON, Stanford University — The work aims at investigating near-wall ignition and flame structure in transverse underexpanded hydrogen jets in high-enthalpy supersonic crossflows generated in an expansion tube. Crossflow conditions are held fixed at $M=2.4$, $p = 40 \text{ kPa}$ and $T \approx 1400 \text{ K}$, while jet-to-crossflow momentum flux ratios $J$ in the range $0.3 - 5.0$ are considered. Schlieren and $OH^*$ chemiluminescence imaging are used to characterize flow structure, ignition and flame penetration, while the instantaneous reaction zone is identified with planar laser-induced fluorescence imaging of OH on side- and plan-view planes. The upstream separation length is found to scale as $J^{0.44} \frac{D}{\lambda}$ (D jet diameter). Similarly, the ignition point $x_{ig}$ strongly depends on $J$: $x_{ig}$ tends to a limiting value of $\sim 22D$ as $J \rightarrow 0$, and the flame is anchored in the upstream recirculation region and lee-side of the jet for $J > 3$. Flame penetration is well described by the traditional form $k \left( \frac{x}{D} \right)^m$ where both $k$ and $m$ are found to depend on $J$ but these parameters reach a limiting value of $k \approx 1$ and $m \approx 0.3$ for $J > 2$. The roles of the unsteady bow shock, the separation and recirculation regions on the near-wall ignition, stabilization and mixing at large $J$ are discussed.

9:57AM MR.00010 Implementation and Validation of a MILES Solver for the Simulation of Supersonic Jet Flow 1, ANDREW CORRIGAN, JUNHUI LIU, K. KAILASANATH, RAVI RAMAMURTI, Naval Research Laboratory — A new MILES (Monotonically Integrated Large-Eddy Simulation) solver has been developed for the simulation of supersonic jet flow and its acoustic properties. The solver uses a finite volume discretization together with the FCT (Flux-Corrected Transport) convection scheme. In order to validate the code, a number of benchmark runs have been performed and compared to those obtained using an FCT solver from the finite element code FEFLO [Liu et al, AIAA, 2009]. The runs use identical input flow conditions and geometry, and include a convergent-divergent nozzle with both over- and under-expanded flow.

Tuesday, November 23, 2010 8:00AM - 10:10AM
Session MS Drops XI Long Beach Convention Center Grand Ballroom A

8:00AM MS.00001 Numerical studies of the deformation of an initially rotating droplet , ERIC POON, ANDREW OOI, University of Melbourne, SHAOPING QUAN, JING LOU, Institute of High Performance Computing, MATTEO GIACOBELLO, Defence Science and Technology Organisation — An initially rotating droplet subjected to an impulsive acceleration by the uniform free stream is studied numerically using the moving mesh interface tracking method (Quan, et al., J. Comp. Phys, 221, 2007) at $Re=40$, $We=40$, $\eta=\lambda=50$. The rotation axis is aligned in the transverse direction and the dimensionless rotation rate, $\Omega^*$, from 0–1 is considered. For $\Omega^* \leq 0.2$, the droplet deforms in a similar fashion to the stationary droplet except the droplet is tilted. At higher $\Omega^*$, the centrifugal force acting on the droplet increases and the droplet is spun radially away from the rotation axis. As a result, the surface area normal to the free stream decreases and this leads to a significant reduction in drag coefficient. The droplet deformation also has a substantial effect on the lift coefficient. As the droplet deforms, the kinetic energy of the rotation is mainly transferred to the surface energy on the interface, which results in a decline in lift coefficient after the initial jump as the surrounding flow field becomes symmetric again.

8:13AM MS.00002 The effect of surfactant redistribution on interactions of deformable drops in gravity and a temperature gradient , MICHAEL ROTHER, University of Minnesota Duluth — Trajectories are calculated by the boundary-integral method for two contaminated deformable drops under the combined influence of buoyancy and a constant temperature gradient at low Reynolds number and with negligible thermal convection. The surfactant is bulk-insoluble, and its coverage is determined by solution of the time-dependent convective-diffusion equation. Two limits are considered. For small drops, the deformation is small, and thermocapillary and buoyant effects are of the same order of magnitude. In this case, comparison is made with incompressible surfactant results to determine when surfactant redistribution becomes important. Convection of surfactant can lead to elimination of saddle points in the relative-trajectory phase plane and can increase the difference between the drops' velocities. For larger drops, deformation can be significant, leading to breakup or capture, and buoyant motion dominates thermocapillary migration. In this case, convection of surfactant can increase deformation and offset previously observed inhibition of breakup for clean drops when the driving forces are opposed.

8:26AM MS.00003 Inertial effects on the dynamics of a drop in a shear flow and the dispersed stresses, RAJESH SINGH, KAUSIK SARKAR, University of Delaware — We numerically simulate the flow field and the streamlines around a drop in shear using a front tracking finite difference method. Inertia destroys the closed streamlines found in a Stokes flow, and gives rise to spiraling and reversed streamlines. Orientation angle of the drop increases with inertia and becomes larger than 45 degrees with increasing inertia at low capillary numbers. However, at higher capillary numbers, it becomes nonmonotonic—it first increases with Reynolds number and then decreases as the drop deformation becomes large. Inertia also introduces transient overshoot and oscillation in the drop deformation before it achieves a steady shape. We provide a simple model for the oscillation that shows the correct scaling with Reynolds and capillary numbers. We also investigate the interfacial tensor that determines the dispersed stress in an emulsion of such drops. We show that the higher than 45 degree inclination leads to a change in sign of the dispersed normal stress differences. The effects of viscosity ratio on the streamline pattern around a drop are also studied.
8:39AM MS.00004 Effect of drop shape on thermally-induced drop motion at the free interface of immiscible liquid layers , EHSAN YAKSHI TAFTI, HYOUNG J. CHO, RANCANATHAN KUMAR, University of Central Florida — Drops at the air interface of immiscible liquids (water on oil) usually form partially-submerged lens shapes. When a lateral thermal gradient is maintained along the surface, such drops move in the direction of decreasing temperatures, as reported in earlier studies. We show that in addition to the lens configuration, it is possible to create spherical (ball-shaped) drops at the interface. Unlike lens-shaped drops, such spherical drops migrate towards warmer regions; i.e. direction of increasing temperatures. Opposite direction of thermally induced motion for drops at the free surface of immiscible liquids is explained based on drop shape and the dynamics of the underlying liquid film subject to a thermal gradient; mainly deformation of the free surface, and the development of an outward moving (hot to cold) flow at the free interface. The proposed physical models predict experimental results satisfactorily. Thermocapillary motion of drops on liquid platforms is ideal for biochemical Microsystems and Lab-on-chip applications where droplets can be transported faster, with higher level of controllability and with less thermal loading of drops as compared to using solid substrates. In addition, other disadvantages of using dry surfaces such as drop evaporation, contamination, and surface pinning are avoided.

8:52AM MS.00005 Transient drop dynamics in convergent-divergent tubes filled with liquids , SHAOPING QUAN, Institute of High Performance Computing — The transient dynamics of a deformable drop with initial momentum moving in convergent-divergent liquid-filled tubes is numerically studied by the moving mesh interface tracking (MMIT)/finite volume method. The geometry effects of the tube on the drop deformation and on the drag coefficient are investigated by varying the radius of the tube neck, and the smallest neck radius is 1.25 times the initial droplet radius. The deformability effects on the droplet dynamics are examined by simulating cases with three Weber numbers. The drop experiences a dramatical deceleration as it approaches and enters the narrow region of the tube, and the drag coefficient increases with the decrease of the radius of the neck. As the Weber number increases, the droplet deforms more, and for the largest Weber number, the initially spherical droplet deforms to a Taylor-drop like shape, especially for the tube with the narrowest neck. After the drop exits the neck, the drop experiences oscillations. The thin film between the drop and the tube wall in the narrow region is resolved by local mesh adaptations. The quantitative analysis will be presented.

9:05AM MS.00006 Effects of Deformation on Drag and Lift Forces Acting on a Droplet in a Shear Flow1, YOUNGHO SUH, CHANGHOON LEE, Yonsei University — The droplet behavior in a linear shear flow is studied numerically to investigate the effect of deformation on the drag and lift acting on droplet. The droplet shape is calculated by a level set method which is improved by incorporating a sharp-interface modeling technique for accurately enforcing the matching conditions at the liquid-gas interface. By adopting the feedback forces which can maintain the droplet at a fixed position, we determine the acting force on a droplet in shear flow field with efficient handling of deformation. Based on the numerical results, drag and lift forces acting on a droplet are observed to depend strongly on the deformation. Droplet shapes are observed to be spherical, deformed, and oscillating for various deformation levels. The method is proven to be applicable to a three-dimensional deformation of droplet in the shear flow, which cannot be properly analyzed by the previous studies. Comparisons of the calculated results by the current method with those obtained from body-fitted methods [Dandy and Leal, J. Fluid Mech. 208, 161 (1989)] and empirical models [Feng and Beard, J. Atmos. Sci. 48, 1856 (1991)] show good agreement.

9:18AM MS.00007 Effect of relative humidity on contact angle of inkjet-printed evaporating colloidal drops , VIRAL CHHASATIA, ABHIJIT JOSHI, YING SUN, Drexel University — The deposition behavior of inkjet-printed aqueous colloidal drops onto glass and polymer (PEN and PET) substrates has been investigated by using fluorescence microscopy, a high-resolution CCD camera, and scanning electron microscopy. Real-time side-view images show that the contact angle of an evaporating colloidal drop is a function of the ambient humidity. The relative humidity also affects the extent to which the drop is able to spread after impacting a substrate, the evaporation rate at the drop surface, and the evaporatively-driven flow inside the drop that drives the suspended particles towards the contact line. The difference between the contact line velocity and liquid velocity at the drop contact line induced by evaporation creates a larger contact angle compared to that of the case without evaporation. This increase in contact angle becomes more significant for a low ambient humidity. Results also show that the particle deposition area and pattern change with the ambient humidity.

9:31AM MS.00008 The electro-hydrodynamic flow about and the shape of strongly deformed drops , EHUD YARIV, DOV RHODES, Technion - Israel Institute of Technology — A liquid drop is suspended in another liquid and is exposed to an otherwise uniform electric field. For strong fields, the drop can elongate significantly. We analyze the strong deformation problem using slender-body analyses, the solution being obtained via expansion in the small slenderness. This parameter is not a priori prescribed, and must be found throughout the course of the method. Comparisons of the calculated results by the current method with those obtained from body-fitted methods [Dandy and Leal, J. Fluid Mech. 208, 161 (1989)] and empirical models [Feng and Beard, J. Atmos. Sci. 48, 1856 (1991)] show good agreement.

9:44AM MS.00009 Contact line dynamics of a liquid meniscus advancing into a microchannel with chemical heterogeneities , CHRISTOPHE WYLOCK, TIPS Department, Universite Libre de Bruxelles, Belgium, MARC PRADAS, Department of Chemical Engineering, Imperial College London, UK, BENOIT HAUT, PIERRE COLINET, TIPS Department, Universite Libre de Bruxelles, Belgium, SERAFIM KALLIADAS, Department of Chemical Engineering, Imperial College London, UK — We examine the motion of a liquid meniscus and associated contact lines advancing into a two-/three-dimensional (2D/3D) microchannel with chemically heterogeneous inner walls. Our study is based on a phase field model of the Cahn-Hilliard type, appropriately modified to take into account the interaction between the fluid and the walls. By solving this model numerically, we characterize the influence of the chemical disorder of the walls on both the interface and contact line dynamics. We perform a detailed statistical analysis of our numerical results by generating several chemical disorder realisations. Examination of the advancing and receding motion of the contact lines in the 2D case reveals that the apparent contact angle suffers a hysteresis behaviour induced by the wall disorder and enhanced as the disorder strength is increased. For the 3D system it is shown that the surface chemical disorder makes the interface and contact line undergo a kinetic roughening process, characterised by a scaling growth with pinning-depinning effects and associated avalanche dynamics.

9:57AM MS.00010 Dynamics of Acoustically Vaporized Microdroplets , ADNAN QAMAR, ZHENG ZHENG WONG, J. BRIAN FOWLKES, JOSEPH BULL, University of Michigan — A combined theoretical and computational approach is utilized to understand the bubble evolution dynamics resulting by vaporizing the superheated dodecafluoropentane (DDFP, C$_2$F$_{12}$) microdroplets via an acoustic perturbation. This work is inspired by a developmental gas embolotherapy technique for cancer treatment by infarcting tumors using selectively formed gas bubbles. The evolution of the bubble comprises of three regimes: an initial linear rapid spherical growth followed by a linear compressed oval shaped growth and finally a slow asymptotic non-linear spherical growth. The bubble evolution process compares quite well with the ultra high-speed experiments. The final bubble radius scales linearly with the initial droplet radius and is approximately five times the initial droplet radius. A pressure pulse with amplitude approximately twice as that of ambient conditions is observed. The pressure pulse wavelength increases with an increasing droplet size whereas the pulse amplitude is weakly dependent on droplet size. This work is supported by NIH grant R01EB006476.
Tuesday, November 23, 2010 8:00AM - 9:57AM
Session MT Biolocomotion X: Macro-Swimming I Long Beach Convention Center Grand Ballroom B

8:00AM MT.00001 Passively pulsed propulsion of aquatic vehicles1, ROBERT WHITTLESEY, JOHN DABIRI, California Institute of Technology — Recent work by Ruiz has shown that pulsed-jet propulsion for aquatic vehicles, similar to that used by sea jellies, salps, and squid, requires a significant decrease in energy input (~30%). These results were obtained despite mechanical inefficiencies in the system to generate the pulsed flow. Thus an approach to generate the pulsed flow using a passive means has been explored. This approach uses collapsible tubing in a pressurized chamber to generate oscillatory, unsteady flow through the outlet. Current results will be presented along with a look toward future developments.

1This research is supported by Office of Naval Research awards N000140810918 and N000141010137.

8:13AM MT.00002 Flapping modes of three filaments placed side by side in a free stream, FANG-BAO TIAN, University of Science and Technology of China, HAOXING LUO, Vanderbilt University, LUODING ZHU, Indiana University–Purdue University Indianapolis, XI-YUN LU, University of Science and Technology of China — Flexible filaments flapping in a surrounding flow are useful models for understanding the flow induced dynamic of soft matter. In this talk we will discuss the dynamics of shed vorticity. The numerical work consists of Euler simulations using FLUENT in which leading edge separation is inhibited. Three kinematic versions of 30000 are investigated experimentally and numerically. In the experimental work, Digital Particle Image Velocimetry (DPIV) is used to examine the strength and dynamics of shed vorticity. Among these patterns, three can be predicted by the linear analysis and have been reported before. These modes are: (1) the three filaments all flap in phase; (2) the two outer filaments are out of phase while the middle one is stable; (3) the two outer filaments are in phase while the middle one is out of phase. The simulations also identified two additional modes: (1) the outer two filaments are out of phase while the middle one flaps at a frequency reduced by half; (2) the outer two filaments are out of phase while the middle one flaps at a slightly different frequency. In addition to the vibratory modes, the drag force and the flapping amplitude are also computed, and the implication of the result will be discussed.

8:26AM MT.00003 Interaction of pitching and heaving flexible flags in a viscous flow, SOHAE KIM, WEI-XI HUANG, HYUNG JIN SUNG, KAIST — In a group of swimming and flying animals, an individual interacts with one another via surrounding flow. Vortices shed by a body are found to strongly influence the downstream body using vortex-vortex and vortex-body interactions. In order to investigate the interactions between flexible bodies and vortices, the present study models two tandem flexible flags in viscous flow by numerical simulation using an improved version of the immersed boundary method. When the downstream flag has pitching and heaving motions, drag on the downstream flag gradually increases and decreases as the pitching and heaving phases vary from 0 to 2π; and the drag coefficient of the downstream flag drops even below the value of a single flag. Such drag variations are influenced by the interactions between vortex shed by the upstream flexible body and vortices surrounding the downstream one. Interaction of tandem flexible flags is investigated as a function of the gap distance between flags, and pitching and heaving phases at intermediate Reynolds numbers. This work was supported by the Creative Research Initiatives (Center for Opto-Fluid-Flexible Body Interaction) and World Class University programs of MEST/NRF.

8:39AM MT.00004 Passive locomotion in unsteady flows, BABAK GHAEMI OSKOUEI, EVA KANSO, University of Southern California — The passive locomotion of a submerged body in unsteady flow is studied. This work is motivated by recent experimental evidence that live and dead trout exploit vortices in the wake of an oscillating cylinder to swim upstream. We consider a simple model of a rigid body interacting dynamically with idealized wake models. The wake models consist of point vortices periodically introduced into the fluid domain to emulate shedding of vortices from an ideal un-deformed fixed or moving obstacle producing a “drag” or “thrust” wake, respectively. Both symmetric and staggered vortex configurations are considered. The submerged body is free to move in the plane, that is to say, it is not pinned at a given point. We do not prescribe a background flow, we rather consider the two-way coupled dynamics between the body’s motion and the advection of ambient vortices. We show that both circular and elliptical bodies could “swim” passively against the flow by extracting energy from the ambient vortices. We obtain periodic trajectories for the body-vortex system and analyze their linear stability. We propose active feedback control strategies to overcome the instabilities.

8:52AM MT.00005 On Gray’s paradox and efficiency measures for swimming, RAHUL BALE, MAX HAO, AMINEET BHALLA, NEELESH PATANKAR, Northwestern University — In 1936 Gray reported that the “drag” power of dolphins was substantially larger than the estimated muscle power. We revisit this “paradox” in the context of undulatory swimming. We consider larval zebrafish as a model system. We question the basic premise of comparing drag power to muscle power. There are two reasons. First, we recognize that it may not be possible decompose the net force on an undulatory swimmer into drag and thrust. If it becomes possible, as we show in our case, the drag power, which represents the work done on the fluid due to motion in the swimming direction, is exactly balanced by the thrust power, which represents the work done by the fluid. Thus, the total power in the swimming direction, computed in this way, is zero. Second, we show that most of the muscle energy is dissipated in causing the lateral motion of the body — not in overcoming the “drag” in the swimming direction. This will be shown based on a power balance equation. Thus, we argue that efficiency measures, that relate the drag power to muscle power, or the Froude efficiency, are not recommended. Instead non-dimensional cost-of-transport could be a useful measure to compare efficiencies of organisms at different scales.

1Supported in part by the NSFC and the NSF.

9:05AM MT.00006 Modeling and numerical simulations of 3D flows past self propelled fishes, MICHEL BERGMANN, INRIA Bordeaux Sud Ouest, ANGELO IOLLO, Institut de Mathématiques de Bordeaux — Modeling and simulation of three-dimensional flows past deformable objects are considered. The incompressible Navier-Stokes equations are discretized in space onto a fixed cartesian mesh. The displacement of self-propelled deforming objects through the fluid is computed from the Newton’s laws (forces and torques computation) and is taken into account using a penalty method. The interface between the solid and the fluid is tracked using a level-set description so that it is possible to simulate several bodies freely evolving in the fluid. The application considered is fish-like swimming. Fish maneuvers and propulsion efficiency for different swimming modes for a single fish or for a fish school are investigated.

9:18AM MT.00007 Hydrodynamics of efficient propulsion in oscillating foils, AZAR ESLAM PANAH, JAMES BUCHHOLZ, The University of Iowa — The flow field and thrust performance of a pitching and heaving NACA 0012 airfoil at a chord Reynolds number of 30000 is investigated experimentally and numerically. In the experimental work, Digital Particle Image Velocimetry (DPIV) is used to examine the strength and dynamic of shed vortical structures. The numerical work consists of Euler simulations using FLUENT in which leading edge separation is inhibited. Three kinematic cases from Anderson et al. (J. Fluid Mech, 360, 1998) are considered, two of which include propulsive efficiency peaks that fall in a Strouhal number range well below that predicted by the stability analysis of Triantafyllou et al. (1991, 1993). By considering the disparate experimental and numerical conditions as well as inviscid model results for these flows in the literature, we will discuss the role of vortex shedding on optimal propulsion.
9:31AM MT.00008 Two-dimensional study of fluid interaction with ray-strengthened fin using immersed boundary method\(^1\). Kourosh ShoelE, Qiang Zhu, UC San Diego — Labriform swimming is a common locomotion mode used by fish in low speed swimming, in which thrust generation is achieved through a combination of flapping and rowing motions of pectoral fins. Bony fishes usually consist of a soft collagen membrane strengthened by embedded flexible rays, rendering anisotropic flexibility. We developed a fluid-structure interaction model based on immersed boundary method to simulate the kinematics and dynamic performance of an idealized 2D fin by considering the flow within one cross-sectional plane. The rays are represented as springs between target points and actual points along the fin, and the membrane is modeled as inextensible beams between the actual points. Using this model we studied thrust generation and propulsion efficiency of the fin at different combinations of parameters. Effects of Reynolds number, flapping frequency as well as different stiffnesses of the rays are studied.

\(^1\)This study was supported by the National Science Foundation under grant CBET-0848857.

9:44AM MT.00009 A Mechanical Fish to Emulate the Fast-Start Performance of Pike. Chengcheng Feng, Yahya Modarres-Sadeghi, University of Massachusetts, Amherst — A northern pike is capable of achieving an instantaneous acceleration of 25g, far greater than that achieved by any manmade vehicle. In order to understand the secrets behind achieving such high accelerations, we have built a mechanical fish to emulate the motion of a pike, a fast-start specialist. A live pike bends its body into a C-shaped curve and then uncoils it very quickly to send a traveling wave along its body in order to achieve high acceleration. We have designed a mechanical fish whose motion is accurately controlled by servo motors, to emulate the fast-start by bending its body to a C-shape from its original straight position, and then back to its straight position. An earlier design of a mechanical fish, which could start from an initial C-shaped curve, shed two vortex rings downstream, resulting in a transfer of energy from the fish to water, and therefore, a reaction force from the fluid to the fish. A maximum acceleration of around 4g was achieved in that design. Our new design adds an additional motion to the sequence by first bending the fish from its straight position into a C-shaped curve. Furthermore, this new mechanical fish is designed to be adjustable in swimming pattern, tail shape, tail rigidity, and body rigidity, making it possible to study the influence of all of these parameters on the fast-start performance.

Tuesday, November 23, 2010 8:00AM - 10:10AM —
Session MU Reacting Flows IV Hyatt Regency Long Beach Regency A

8:00AM MU.00001 Magnetohydrodynamic Augmentation of Pulse Detonation Engines\(^1\). Christopher Zeineh, Lord Cole, Ann Karagozian, University of California, Los Angeles — Pulse detonation engines (PDEs) are the focus of increasing attention due to their potentially superior performance over constant pressure engines. Yet due to its unsteady chamber pressure, the PDE system will either be over- or under-expanded for the majority of the cycle, with energy being used without maximum gain. Magnetohydrodynamic (MHD) augmentation offers the opportunity to extract energy and apply it to a separate stream where the net thrust will be increased. With MHD augmentation, such as in the Pulse Detonation Rocket-Induced MHD Ejector (PDRIME) concept, energy could be extracted from the high speed portion of the system, e.g., through a generator in the nozzle, and then applied directly to another flow or portion of the flow as a body force. The present high resolution numerical simulations explore the flow evolution and potential performance of such propulsion systems. An additional magnetic piston applying energy in the PDE chamber can also act in concert with the PDRIME for separate thrust augmentation. Results show that MHD can indeed influence the flow and pressure fields in a beneficial way in these configurations, with potential performance gains under a variety of flight and operating conditions. There are some challenges associated with achieving these gains, however, suggesting further optimization is required.

\(^1\)Supported by AFOSR.

8:13AM MU.00002 Simulations of a Detonation Wave in Transverse Magnetic Fields\(^1\). Lord Cole, Ann Karagozian, University of California, Los Angeles, Jean-Luc Cambier, Air Force Research Laboratory — Numerical simulations of magnetohydrodynamic (MHD) effects on detonation wave structures are performed, with applications to flow control and MHD power extraction in Pulse Detonation Engines (PDE) and their design variations. In contrast to prior studies of MHD interactions in PDE\(^1\), the effects of the finite relaxation length scale for ionization on the stability of the detonation wave are examined. Depending on the coupling parameters, the magnetic field can quench the detonation and effectively act as a barrier to its propagation. Conversely, an applied transient magnetic field can exert a force on a pre-ionized gas and accelerate it. The dynamics are subject to non-linear effects; a propagating transverse magnetic field will initially exert a small force if the gas has a low conductivity and the magnetic Reynolds number \(Re_m\) is low. Nevertheless, the gas accelerated by the “piston” action of the field can pre-heat the ambient gas and increase its conductivity. As the wave progresses, \(Re_m\) increases and the magnetic field becomes increasingly effective. The dynamics of this process are examined in detail with a high-order shock-capturing method and full kinetics of combustion and ionization. The complex chemical kinetics calculations are ported onto a GPU using the CUDA language, and computational performance is compared with standard CPU-based computations.

\(^1\)Cambier, et al., AIAA-2008-4688

8:26AM MU.00003 RANS Simulations of Supersonic Combustion using a Flamelet-based Model\(^1\). Vincent Terrapon, Rene Pecnik, Frank Ham, Heinz Pitsch, Stanford University — A flamelet-based model for supersonic combustion is introduced. Since viscous heating and compressibility effects play an important role in high-speed flows, the flamelet implementation originally based on a low Mach number assumption has been reformulated. In this new implementation temperature is not any longer given by a chemistry table but computed from the total energy and the tabulated species mass fractions. Additionally, the source term in the progress variable transport equation is rescaled by the pressure to better account for compressibility effects. This approach allows the use of complex chemistry with only 2 or 3 additional scalar transport equations. The model is applied to a RANS simulation of a hydrogen jet in a supersonic crossflow and the results are compared with experimental measurements. Finally, the model is also used in the RANS computation of the hydrogen fueled HyShot II scramjet and simulation results are compared with experimental data from a ground experiment.

\(^1\)This material is based upon work supported by the Department of Energy [National Nuclear Security Administration] under Award Number(s) DE-FC52-08NA28614.
8:39AM MU.00004 The Dynamics of Unsteady Detonation with Diffusion, CHRISTOPHER ROMICK, University of Notre Dame, TARIQ ASLAM, Los Alamos National Laboratory, JOSEPH POWERS, University of Notre Dame. — We consider an unsteady one-dimensional detonation with diffusion. The system studied is a standard one-step kinetics model whose inviscid stability properties are well characterized. The introduction of diffusion creates an interaction between the length scales of reaction and diffusion thus delaying the onset of instability of the system when the length scales of diffusion and reaction overlap. This interaction is admitted in systems of complex kinetics where the finest reaction length scales are comparable to those of a viscous shock.

1Department of Aerospace and Mechanical Engineering
2Dynamic and Energetic Materials Division
3Department of Aerospace and Mechanical Engineering

8:52AM MU.00005 A multivariate quadrature based approach for LES based supersonic combustion modeling1, PRATIK DONDE, HEESEOK KOO, VENKAT RAMAN, The University of Texas at Austin — The direct quadrature method of moments (DQ MOM) was developed to solve high-dimensional probability density function (PDF) equations that arise in the description of turbulent combustion. This method is particularly useful in shock-containing supersonic internal flows such as those encountered in scramjet engines. In the DQ MOM approach, the PDF is described in terms of a finite number of weighted delta functions whose weights and locations in composition space are obtained by solving specific transport equations. Since this approach is fully Eulerian in nature, it is advantageous compared to conventional Lagrangian methods used for solving the PDF transport equation. However, implementation of this formulation in the context of the large eddy simulation (LES) methodology leads to large numerical errors. For instance, the high-resolution numerical schemes used in LES lead to non-realizable and diffusive evolution of the DQ MOM equations. Here, we propose a novel semi-discrete quadrature method of moments (SeQMOM) that overcomes this problem. A decoupling procedure is used to extend this method to multivariate PDF descriptions. The numerical implementation in LES as well as validation exercises will be presented.

1We thank NASA (Grant: NNX08AB41A) for supporting our research.

9:05AM MU.00006 Large Eddy Simulation of Supersonic Combustion1, AMIRREZA SAGHAFIAN, VINCENT TERRAPON, FRANK HAM, HEINZ PITSCH, Naval Research Laboratory — Large eddy simulation of supersonic combustion is performed based on a flamelet/progress variable combustion model. This model was originally developed for low Mach number, where temperature and species mass fractions are looked up from a pre-computed flamelet library. In the compressible formulation presented here, the equation for the total energy is solved to find temperature. Because total energy is a non-linear function of temperature, an iterative method like Newton-Raphson is inevitable. However, a new formulation is introduced to eliminate this expensive iterative step. Large eddy simulation of under-expanded hydrogen jet in supersonic cross-flow is performed and results are compared with experiments. For sufficiently high jet to cross-flow momentum ratio, burning of the fuel is observed in the upstream region of the jet exit and the length of this burning region is in good agreement with experimental data. In addition, reaction is also observed in a large portion of the boundary layer downstream of the jet consistent with experimental observations.

1This work is supported by the US Department of Energy under the Predictive Science Academic Alliance Program (PSAAP) Program.

9:18AM MU.00007 The Interaction of High-Speed Turbulence with Flames: Turbulent Flame Speed, ALEXEI POLUDNENKO, ELAINE ORAN, Naval Research Laboratory — The interaction of flames with background turbulence occurs in systems ranging from chemical flames on Earth to thermonuclear burning fronts in supernovae. We present an analysis of a set of numerical simulations aimed at studying the dynamics and properties of turbulent flames formed under the action of high-speed turbulence in stoichiometric hydrogen-air mixture. The simulations were performed using the massively parallel reactive-flow code Athena-RFX. Previous analysis of these simulations showed that this system represents turbulent combustion in the thin reaction zone regime even in the presence of intense turbulence (Da = 0.05, U_in, ∼ 35 times the laminar flame speed). Here we discuss the processes that determine the turbulent burning velocity and show that it exceeds values that can be attributed only to the increase of the flame surface area. We suggest a possible mechanism for this excess burning rate. Finally, we discuss the implications of these results for the process of deflagration-to-detonation transition in unconfined systems. This work was supported in part by AFOSR, NRL, ONR, and by NSF through the TeraGrid resources.

9:31AM MU.00008 Compressible, diffusive, reactive flow simulations of the double Mach reflection phenomenon, J.L. ZIEGLER, California Institute of Technology, R. DEITERDING, Oak Ridge National Laboratory, J.E. SHEPHERD, D.I. PULLIN, California Institute of Technology — We describe direct numerical simulations of the multi-component, compressible, reactive Navier-Stokes equations in two spatial dimensions. The simulations utilize a hybrid, WENO/centered-difference numerical method, with low numerical dissipation, high-order shock-capturing, and structured adaptive mesh refinement (SAMR). These features enable resolution of diffusive processes within reaction zones. A series of one- and two-dimensional test problems are used to verify the simulation, specifically the high-order accuracy of the diffusion terms, including a viscous shock wave, the decaying Lamb-Oseen vortex, laminar flame and unstable ZND detonation. High-resolution simulations are discussed of the reactive double Mach reflection phenomenon. The diffusive scales (shear/mixing/boundary layers and flame thicknesses) and weak shocks are resolved while the strong shocks emanating from the triple points are captured. Additionally, a minimally reduced chemistry and transport model for hydrocarbon detonation is used to accurately capture the induction time, chemical relaxation, and the diffusive mixing within vortical structures evolving from the triple-point shear layer.

9:44AM MU.00009 Thermomechanical phenomena in rapidly heated inert and reactive gases1, DAVID KASSOY, University of Colorado, Boulder — Asymptotic methodologies are used to identify thermomechanical processes occurring in inert and chemically reactive gases subsequent to deposition of transient, spatially resolved thermal energy into a finite volume (“near-field”) of inert or chemically reactive gases. Rational models are developed for a wide range of energy depositions, heating time scales and volume dimensions. When the Mach number of the gas expelled from the heated volume boundary is small, only linear acoustic disturbances can appear in the neighboring, unheated gas (“far-field”). Larger boundary Mach numbers are associated with shock waves in the “far-field.” An extreme example describes a spatially resolved heated source for extremely strong blast waves associated with nuclear explosions. The classical model for strong blast waves is reformulated to provide a physically sound explanation of the singularities in the well-known similarity solutions used to describe blast wave evolution. A model for the evolution of a reaction center (hot spot) is considered to identify the characteristic gasdynamics generated by rapid heat addition.

1Elements of this work have been supported by AFOSR.
9:57AM MU.00010 Hot Surface Ignition and Flame Propagation of Hydrocarbon Air Mixtures, PHILIPPE BOETTCHER, BRIAN VENTURA, GUILLAUME BLANQUART, JOSEPH SHEPHERD, California Institute of Technology — To mitigate the risk of accidental explosions in industrial facilities and in the aviation industry, the mechanisms and parameters leading to ignition must be investigated. Of particular interest are isolated hot surfaces in contact with gaseous hydrocarbon fuels, and thus ignition of premixed n-hexane and n-heptane air mixtures is examined using a high temperature glow plug. Measurements include schlieren visualization, particle streaks, pressure, and temperature measurements in the plume created by the hot surface. These measurements are performed for experiments in both air and combustion mixtures ranging in equivalence ratio from 0.5 (near the lower flammability limit) to 3.0. This allows for comparison of ignition temperature, flame speed, pressure rise, and temperature distribution with a computational flame model. For equivalence ratios above 0.7 the ignition temperature was observed to be insensitive to increasing fuel concentration and showed good agreement with the model. Three distinct combustion modes are observed that scale with the Richardson number: single flame, multiple flames, and puffing. These behaviors show the transition from flame propagation dominated to buoyancy dominated behavior, with puffing cycles of the order 10 Hz.

Tuesday, November 23, 2010 8:00AM - 9:57AM – Session MV Suspensions I Hyatt Regency Long Beach Regency B

8:00AM MV.00001 Microstructure in Concentrated Sheared Dispersions, JEFF MORRIS, EHSAN NAZODKLAST, City College of New York — This work describes a theory for predicting microstructure of concentrated colloidal hard spheres as a function of Péclet number $Pe = \frac{\gamma a}{\gamma_{a}}$ and particle volume fraction, $\phi$, $\gamma$ is the shear rate, $a$ is the particle radius, $\gamma$ is fluid viscosity and $kT$ is the thermal energy. We study the pair distribution using the pair Smoluchowski equation. Many-body effects in the conservation equation were then formulated self-consistently through probabilistic third-particle integrals, with emphasis on capturing the interaction of flow and excluded volume effects. The resulting integro-differential equation was solved iteratively. Comparison between theory predictions and simulation results show that the theory is able to predict known near-equilibrium ($Pe \ll 1$) and dilute-suspension large-$Pe$ results. The approach accurately predicts the major features of microstructure at concentrated $\phi$ under strong shear, which differentiates it from previous theoretical work. Rheological quantities of shear stress, normal stress differences, and particle pressure are computed from the structure.

8:13AM MV.00002 Bulk Rheology of Noncolloidal Deformable Fiber Suspension, MUBASHAR KHAN, MINH DO-QUANG, GUSTAV AMBERG, KTH Royal Institute of Technology, JINGSHU WU, CYRUS AIDUN, Georgia Tech — The effect of fiber flexibility and shear rate on microstructure and rheology of fibers suspended in Newtonian fluid is investigated with direct numerical simulation based on the external boundary force lattice-Boltzmann method (Wu and Aidun, *Int. J. Multiphase Flow*, 36 (3) March 2009). It is shown that the fiber bending ratio (BR), which is proportional to the fiber elastic modulus divided by dynamic viscosity and shear rate, has significant influence on rheology. For fiber suspension under shear, the relative viscosity decreases significantly as BR increases. The primary normal stress difference undergoes a minimum at a critical BR value and increases rapidly as BR decreases further (Wu and Aidun, *J. Fluid Mech.* to appear 2010). In previous studies, the effect of shear rate on bulk rheology at a constant BR and volume fraction has remained unclear. In this study, we show that the ratio of viscosity for deformable (BR=0.25) to rigid fibers for volume fraction 0.05 may reach a maximum at a critical shear rate. We explain this behavior in terms of fiber orientation distribution and fiber-fiber interaction.

8:26AM MV.00003 Normal stresses in non-Brownian suspensions: bulk rheology and particle migration, FRANCOIS BOYER, OLIVIER POULIQUEN, ELISABETH GUAZZELLI, IUSTI, CNRS - Aix-Marseille Université — Concentrated suspensions are known to exhibit non-Newtonian effects and classical rheology often fails to give a consistent description of actual flows. Particle migration has been shown to be the main reason for both thixotropic effects and inhomogeneous microstructure of sheared suspensions, and is thus of primary interest for applications. The suspension balance model relates the particle migration to the suspension normal stresses and is therefore a consistent set of constitutive equations. Recently, this model has been revisited and other mechanisms have been proposed. In that context, an experimental study that characterize both normal stresses and particle migration independently is of great interest. We report measurements of normal stress differences in concentrated suspensions of non-Brownian spheres thanks to a rotating-rod rheometer. Normal stresses are experimentally shown to vanish below a critical volume fraction of 0.22: this could therefore indicate a non-hydrodynamic origin of macroscopic normal stresses. Furthermore, the time-dependent behaviour of the system is the signature of particle migration: comparisons with theoretical predictions allows to test quantitatively particle migration models.

8:39AM MV.00004 Experimental study of structure and rheology of concentrated colloidal suspensions, XIANG CHENG, Dept. of Physics, Cornell University, JONATHAN MCCOY, Dept. of Physics, Colby College, JACOB ISRAELAVI, Dept. of Chemical Engineering, Univ. of California, Santa Barbara, ITAI COHEN, Dept. of Physics, Cornell University — We investigate the three-dimensional flow structure and rheology of concentrated colloidal suspensions in a newly designed plane shear cell. Using fast confocal microscopy, we optically probe the dynamics of colloidal suspension under shear with single particle resolutions. The rheological properties of the suspensions are also measured simultaneously. Hence, a direct correlation between the microstructure and the rheology of the suspensions is obtained in our experiments. Three regimes are observed with increasing shear rate. At low shear rates where the Brownian motion of particles dominates, the structure of the suspensions is indistinguishable from that at equilibrium. At intermediate shear rates, colloidal particles form sliding layers normal to the shear gradient direction, but keep a disordered structure within the layers. Along with this transition, the suspensions shear thins dramatically. At even higher shear rates, particles in the layers organize into a novel structure – strings of particles perpendicular to the shear direction. We speculate that this structure may be a precursor to the hydroclusters observed in shear thickening suspensions.

8:52AM MV.00005 Irreversibility and Chaos: Role of Long Range Hydrodynamic Interactions in Sheared Suspensions, BLOEN METZGER, IUSTI-CNRS UMR 6595, JASON E. BUTLER, The University of Florida — Non-Brownian particles suspended in an oscillatory shear flow are studied numerically. In these systems it is often assumed that chaos (due to the long-range nature of the hydrodynamic interaction between particles) plus noise (contact or roughness) lead to irreversible behavior. However, we demonstrate that the long-range hydrodynamic interactions are not a source, nor even a magnifier, of irreversibility when coupled with non-hydrodynamic interactions.

1Partial Support from the Gunnar Nicholson Faculty Exchange Fellowship is acknowledged.

3We acknowledge the ANR JCJC SIMI 9 for supporting this collaboration.
9:05AM MV.00006 Direct measurement of a normal stress in a sheared suspension1. JEROME MARTIN, CNRS, Lab. FAST, Orsay, FR, GEORGES GAUTHIER, STEPHEN GARLAND, University Paris XI, Lab. FAST, Orsay, FR, ANGELIQUE DEBOEUF, ESPCI, Lab. PPM, Paris, FR, JEFFREY MORRIS, Levich Institute, CCNY, LAB. FAST, ORSAY, FR TEAM, PR JEFFREY MORRIS COLLABORATION — A method was recently proposed (Deboeuf et al., Phys. Rev. Lett., 2009), to measure the shear-induced “particle pressure” in a sheared non-colloidal suspension. The ‘particle pressure’ was obtained in a Couette device through the liquid pressure, measured behind a grid permeable to the fluid but impermeable to the particles, placed at the outer cylindrical wall of the device. The liquid pressure is equal to the (vertical) component of the particle stress, in the direction of the vorticity. It gives a good estimation of the particle pressure, assuming the shear-induced particle stress is nearly isotropic. The apparatus enables also the measurement of the total pressure at the outer wall. Coupled with the grid pressure, the latter measurement gives access to the radial component of the particle stress. Our collected data demonstrate that anisotropy does exist, with a normal stress one order of magnitude lower than the particle pressure.

9:18AM MV.00007 Rolling and resuspension of a particle for explanation of Shields diagram. HYUNGOO LEE, S. BALACHANDAR, University of Florida — Recent research (Zeng et al. 2009, Lee & Balachandar 2010) has shown that both the shear-and wall-induced lift contributions on a particle sharply increase as the gap between the wall and the particle is decreased. Explicit expressions that are valid over a range of finite Re were obtained for the drag and lift forces in the limiting cases of a stationary particle in wall-bounded linear flow and of a particle translating parallel to a wall in a quiescent ambient. Here we consider the more general case of a translating and rotating particle in a wall-bounded linear shear flow where shear, translational and rotational effects superpose. We have considered a modest Reynolds number range of 1-100. Direct numerical simulations using immersed boundary method were performed to systematically figure out the characteristics of hydrodynamic forces on a finite-sized particle moving while almost in contact with a wall. We present composite correlation for the hydrodynamic forces which are in agreement with all the available low-Reynolds number theories. Also, obtained lift, drag and torque are used to explain the Shields diagram.

9:31AM MV.00008 Instability of settling non-spherical particle in a vertical shear flow, DEWEI QI, Western Michigan University, DONALD KOCH, Cornell University, GANESH SUBRAMANIAN, JNCASR, Bangalore, India — Two mechanisms are attributed to the cross-stream migration when fiber settles in a vertical shear flow. First, a particle may migrate toward streamlines of the imposed shear flow with smaller downward fluid velocities, due to relative translation of the particle and fluid, called the Saffman effect. Second, a non-spherical particle at finite Reynolds number will attempt to rotate with its long body along the horizontal direction due to inertial torque. On the other hand, the torque due to the imposed vertical shear flow rotates the non-spherical in the opposite direction. The dynamic balance between the two torques can lead to a small angle between the particle long body and horizontal plane and may drive the particle migrate toward the streamlines of the shear flow with the large downward fluid velocity. The second mechanism was recently proposed by Shin, Koch and Subramanian. A fiber with aspect ratio $\kappa = 2, 1.6, 1.2, 1.1$ and 0 is used to study the lateral migration. It is shown that at a given shear and aspect ratio, fiber lateral migration can be divided into three phases depending on the Reynolds number. The simulation results identified the lateral migration phase diagram and confirm the second mechanism.

9:44AM MV.00009 Anomalous diffusion of non-colloidal suspensions in a Couette flow. KYONGMIN YEO, MARTIN R. MAXEY, Division of Applied Mathematics, Brown University — The effects of wall-confinement on the dynamics and particle migration within concentrated non-colloidal suspensions in a Couette flow are investigated. We focus mainly on the shear-induced self-diffusion at 40% volume fraction. The channel is divided into four zones depending on suspension microstructures and variances of the wall-normal and spanwise displacements for the particles in each zone are studied. Due to the strong spatial coherency, the suspended particles exhibit anomalous diffusion in the wall-normal direction. The diffusive behavior changes from superdiffusion for the particles next to the wall to subdiffusion for the particles near the core of the channel. The results indicate that the intermittent jumps and particle entrapment in particle layers are responsible for the anomalous diffusion near the wall, while the subdiffusion in the core is related with the restriction on the available lengthscale by the size of confinement. Diffusive behaviors of the particles in the core of the channel for four different volume fractions (25%, 30%, 35%, and 40%) are compared. For a channel of height $20D$, where $D$ is the particle diameter, the regular diffusive behavior in the core is observed for the volume fraction 40%.

Tuesday, November 23, 2010 8:00AM - 10:10AM — Session MW Experimental Techniques II Hyatt Regency Long Beach Regency C

8:00AM MW.00001 Laser imaging measurements of flow dynamics and mixing in gel-phase flows, LESTER K. SU, JASON P. LEGGETT, Johns Hopkins University, MILLICENT A. COIL, Orbital Technologies Corp — Gelled hypergolic propellants are interesting in rocket propulsion applications, in combining the stability of solid propellants and the controllability of liquid propellants. To exploit these advantages fully, we require an improved understanding of the flow and mixing properties of gel-phase fluids. In this work, we apply planar laser-induced fluorescence (PLIF) to investigate gel mixing in a mixing layer geometry, and particle image velocimetry (PIV) to measure flow velocities in and around two impinging gel streams. We consider two water-based gels (Ultraz 10) and mineral-oil-based gels (Kraton G1650) of varying compositions (strengths). For the PLIF, these gels are doped with disodium fluorescein dye. We will discuss some of the issues attendant to the application of these laser diagnostic methods in the gel phase, and we will illustrate how these gel-phase flows differ from flows of Newtonian fluids in similar flow geometries.

8:13AM MW.00002 Low Pressure Seeder Development for PIV in Large Scale Open Loop Wind Tunnels, RYAN SCHMIT, Air Force Research Laboratory — A low pressure seeding techniques have been developed for Particle Image Velocimetry (PIV) in large scale wind tunnel facilities was performed at the Subsonic Aerodynamic Research Laboratory (SARL) facility at Wright-Patterson Air Force Base. The SARL facility is an open loop tunnel with a 7 by 10 foot octagonal test section that has 56% optical access and the Mach number varies from 0.2 to 0.5. A low pressure seeder was designed and tested in the inlet of the wind tunnel. The seeder was designed to produce an even distribution of seed across the test section. V/Count Compact 5000 using Smoke Oil 180 was using as the seeding material. The results show that this low pressure seeder does produce steady seeding but excellent PIV images are produced.

8:26AM MW.00003 Large depth-of-field PIV in a narrow channel, DANA EHYAEL, KENNETH KIGER, University of Maryland — The current work is motivated by a goal to obtain quantitative temporally-resolved velocity measurements of buoyant natural convection within a Hele-Shaw cell. In contrast with typical micro-PIV studies, PIV in a Hele-Shaw cell requires a large field-of-view in comparison to the channel gap spacing, precluding the use of a thin light sheet or a small depth-of-field that can isolate a narrow region of the local Poiseuille velocity profile across the gap. This necessitates imaging particles across the whole depth, causing the cross-correlation to become broadened by the velocity gradients across the gap. In addition to the velocity gradients, the finite Reynolds numbers associated with typical flow conditions may cause significant inertial migration of the seed particles, creating an evolving and non-uniform concentration distribution, which in turn will change the shape and relative peak location of the cross-correlation. In order to make a quantitative relationship to local mean flow within the gap, a uniform flow is studied experimentally and modeled using synthetic image generation at various positions along the flow. This information will then be used to optimize the conditions for reliable PIV interrogation, and sample results of buoyant convection will be given.
8:39AM MW.00004 Particle shadow velocimetry vs. LDV: measurements of a turbulent pipe flow. M. McPHEAL, J. GIORDANO, A. FONTAINES, M. KRANE, ARL Penn State, L. GORS, J. GRAFTON. Innovative Scientific Solutions, Inc. — Image pre-processing was used to improve multi-color Particle Shadow Velocimetry (PSV) measurements of a near-wall turbulent pipe flow. These included corrections for color cross-talk, color aberration and image distortion. Multi-color PSV is a modification of DIPV which employs pulsed multicolor lights for illumination. The particle shadows are imaged to produce 2-D vector fields with traditional DPIV correlation methods. Multi-color PSV allows for higher temporal sampling rates than conventional DPIV, at lower cost. Aberrations were quantified by imaging a target with known geometry and generating a mapping function to correct the multicolor shadows for chromatic aberration shadow displacement shifts between colors. PSV images were spatially corrected with these functions. Color cross-talk was corrected for by subtracting average cross-talk intensity from the local intensity. Corrected mean velocity and second order statistics for turbulent pipe flow in the Penn State ARL glycerin tunnel showed favorable comparison to LDV and standard PIV measurements.

1 Acknowledge support from PSU-ARL Water Tunnel summer intern program.

8:52AM MW.00005 A vision-based hybrid particle tracking velocimetry (PTV) technique using a modified cascade-correlation peak-finding method. W. TIEN, Y. LEI, J. DUNCAN, D. DABIRI. University of Washington, T. ROSEN, ETH Zurich, J. HOVE, University of Cincinnati, M. GHRIB, California Institute of Technology. — In this talk we present new algorithms for particle identification and particle tracking velocimetry (PTV). The new particle identification algorithm uses both the Cascade Cross-correlation Method (CCM) and 2-D surface Gaussian fitting to overcome the issue of overlapping particles. Simulation with up to 5% noise shows particles can be located with errors under 0.07 pixels for overlap ratios up to 50%. The new PTV method, taking advantage from vision theory, is developed to map from a “proximity” matrix to a “pairing” matrix while inherently satisfying the exclusion principle. The validity of the results is ensured by hybridization with PIV data, and the reliability of the method is further tested by adding noise to synthetic data. The proposed method gives reliability values up to 99.9% at particle densities up to 0.06 for a simulated moving wall flow, an oscillating wall flow and an Oseen vortex. Variations in particle intensity and diameter are also tested with simulated flow images. An experimental shear layer image pair is then tested, proving the robustness of the proposed method. Furthermore, it is shown that kriging interpolation in combination with PTV results can accurately resolve velocity gradients such as the wall region.

9:05AM MW.00006 A compact Self-Contained Underwater Velocimetry Apparatus (SCUVA) for in situ field measurements in daytime. M. KINZEL, J. DABIRI, Caltech. — In-field measurements at remote location present a challenge for the measurement systems involved. Not only do these systems have to be self-sufficient in regard to power supply and data acquisition but also robust and easy to handle. With the current level of miniaturization in electronics it becomes possible to construct PIV-systems, which meet these criteria and are even small enough to be used as hand held devices. Following the work of Katija and Dabiri, we present a PIV-system, which is designed for SCUBA divers to take in-field measurements of the flow around marine organisms in daytime. The fact that the system can be operated in daytime makes work for the divers considerably easier. On the other hand it presents an additional challenge due to the laser power, which can be installed in portable devices. A detailed description of the measurement setup will be given together with a discussion of some preliminary results.

9:18AM MW.00007 3D Synthetic Aperture Imaging for Fluid Flows. J. BELDEN, MIT, T. TRUSCOTT, B. AXIAK, A. TECHET, MIT. — Three-dimensional and multiphase fluid flow environments demand advanced and innovative measurement systems in order to fully resolve the flow physics. We present implementations of synthetic aperture imaging techniques for 3D particle image velocimetry and bubble flow field extraction. This work lays the foundation for a comprehensive tool for measuring multiphase flows. Simulations have shown 3D synthetic aperture particle image velocimetry (SAPIV) to be a promising technique for resolving 3D velocity fields in densely seeded flows using an array of cameras. Here, we experimentally study a canonical vortex ring using a low-cost 8 camera array and benchmark the results with standard 2D PIV. Also, using a high speed camera array, we apply synthetic aperture imaging to the 3D bubble field entrained by a jet impinging on a free-surface.

9:31AM MW.00008 Modern quantitative schlieren techniques. M. HARGATHER, G. SETTLES, Pennsylvania State University. — Schlieren optical techniques have traditionally been used to qualitatively visualize refractive flowfields in transparent media. Modern schlieren optics, however, are increasingly focused on obtaining quantitative information such as temperature and density fields in a flow — once the sole purview of interferometry — without the need for coherent illumination. Quantitative data are obtained from schlieren images by integrating the measured refractive index gradient to obtain the refractive index field in an image. Ultimately this is converted to a density or temperature field using the Gladstone-Dale relationship, an equation of state, and geometry assumptions for the flowfield of interest. Several quantitative schlieren methods are reviewed here, including background-oriented schlieren (BOS), schlieren using a weak lens as a “standard,” and “rainbow schlieren.” Results are presented for the application of these techniques to measure density and temperature fields across a supercritical turbulent boundary layer and a low-speed free-convection boundary layer in air. Modern equipment, including digital cameras, LED light sources, and computer software that make this possible are also discussed.

9:44AM MW.00009 Megahertz rate Schlieren visualization of underexpanded, impinging jet using pulsed high power LED. C. WILLERT, W. GILBERT, R. KELLER, D. MITCHELL, J. SORIA, L. LTRAC, Monash University. — Recent advances in light emitting diode (LED) technology has resulted in high power, single chip devices that provide luminous radiant fluxes exceeding several watts. Operated in pulsed current mode the instantaneous light emission of an LED can be further increased to levels comparable to that of photographic (xenon) flash units making it a suitable light source for Schlieren imaging. Compared to the commonly used xenon flash units an LED can be triggered within tens of nanoseconds at rise times on the order of 100 ns thereby enabling stroboscopic illumination at megahertz rates. In the present application the LED’s driving electronics were synchronized to a high speed camera to provide time-resolved Schlieren images of an underexpanded free jet impinging on a flat plate (nozzle pressure ratio 2.0 to 5.2). The LED was pulsed in burst mode for 102 images at currents of up to 120 A at 500 ns per pulse. Compared to images obtained with a xenon white light flash the nearly monochromatic green light of the LED results in much crisper flow features with superior repeatability in intensity without any speckle artifacts commonly found with laser illumination.
9:57AM MW.00010 Direct Measurement of Turbulent Shear\footnote{This work is supported by NSF grant No. DMR 0604477.} . STEFANUS STEFANUS, STANLEY STEERS, WALTER GOLDBURG, University of Pittsburgh — Photon Correlation Spectroscopy (PCS) is used to directly measure the mean shear rate $\tau$ in a turbulent soap film. A 5 mW 633 nm He-Ne laser is focused on the film at a point $r$, the spot size being $w=100 \mu m$. The scattered light intensity $I(t)$ is analyzed by a correlator that measures the average, over time $t$, of the correlation function $G(\tau) = \langle I(t)I(t+\tau)/\langle I(t)\rangle^2 - 1$. From $G(\tau)$, one extracts the shear $\tau$ averaged over $w$ and the standard deviation of $\tau$. Of special interest is the shear at points $r$ near a solid boundary. The PCS measurements of $\tau$ (in Hz) are compared with those obtained by laser Doppler velocimetry (LDV). The two techniques yield values of $\tau$ that agree within a standard deviation. The PCS method has the advantage of compactness and rapid data collection, making it of potential use in biology and medicine. By changing the orientation of the incident and scattered beams, one can measure various components of the shear tensor. The implementation of the PCS method does not require the presence of a mean flow. It can also be applied to three-dimensional turbulence.

Tuesday, November 23, 2010 8:00AM - 10:10AM –
Session MX Industrial Applications I Hyatt Regency Long Beach Regency D

8:00AM MX.00001 Aerodynamic stability of blunted-cone heat shields for atmospheric entry vehicles , JOHN SADER, ELEANOR BUTTON, DANIEL LADIGES, CHARLES LILLEY, NICHOLAS MACKENZIE, EDWARD ROSS, The University of Melbourne — Spacecraft entry into the atmosphere of a planet requires protection against the extreme temperatures that result from aerodynamic heating. This is normally achieved through use of a heat shield, which also provides the necessary aerodynamic braking and stability. The shape of the heat shield used varies considerably between spacecraft, and spherical and blunted-cone geometries are often employed. The “blunted-cone” heat shield has been developed through experimental design and computational simulation. Here, we demonstrate that this generic shape can be derived mathematically and yields the maximum stabilizing aerodynamic torque of all possible shapes. The derived single shape is universal, depending only on the center-of-mass, and provides invariance in static stability due to minor heat shield damage. Importantly, the design of practical heat shields involves numerous competing factors, which include the expected heat load and the craft volumetric efficiency, in addition to aerodynamic stability. We thus emphasize that the presented results focus on only one component of this multi-objective problem. Nonetheless, the derived shape shows good agreement with the heat shields of previous entry vehicles, a comparison of which shall be given.

8:13AM MX.00002 An Experimental Investigation of Compressible Dynamic Stall on a 2-D Airfoil Subjected to Non-Harmonic Pitching Motion , DUSTIN COLEMAN, FLINT THOMAS, THOMAS CORKE, PATRICK BOWLES, KATIE THORNE, University of Notre Dame — An experimental study of dynamic stall was conducted on a 2-D airfoil operating at Reynolds number up to $2\times10^6$ and over a Mach number range of 0.2 - 0.4. The primary pitching frequency was 6.58 Hz producing reduced frequencies ranging from 0.035 - 0.075. The facility was constructed to allow a second pitching frequency to be mechanically added to the primary mode of the airfoil providing both harmonic and non-harmonic disturbances to the pitch motion in order to simulate aerodynamic conditions where stall flutter may occur. Static pressure data was acquired using 30 surface mounted dynamic pressure transducers simultaneously sampled at a rate of 5 kHz. Instantaneous pressure time series and resultant forces were analyzed to elucidate the consequent fluid interactions. Preliminary results indicate that the higher frequency input provides a mechanism for increased aerodynamic stability throughout the dynamic stall cycle.

8:26AM MX.00003 Compressibility effects on dynamic stall attributes , PATRICK BOWLES, THOMAS CORKE, FLINT THOMAS, KATIE THORNE, DUSTIN COLEMAN, University of Notre Dame — An experimental study of the compressibility effects on the load, stability, and separation characteristics of a modern rotor-blade geometry is presented under dynamic stall conditions. The airfoil was oscillated in pitch about the quarter chord at freestream Mach numbers from 0.2 to 0.55, reduced frequencies from 0.025 to 0.10, and Reynolds numbers up to 3.5 million. All values relevant to a helicopter retreating or advancing blade. Thirty high frequency absolute pressure transducers measured the airfoil’s static pressure distribution. Emphasis was placed on the development of the leading edge vortex. Increased freestream Mach numbers resulted in a reduced ability of the near leading edge flow to overcome the adverse pressure gradient on the airfoil’s suction side, limiting dynamic load overshoot as well as negative damping. Indeed, light dynamic stall conditions showed a greater aptitude to toward unstable motion. At free-stream Mach numbers greater than 0.4, airfoil local Mach numbers consistently near 1.6 prior to an abrupt flow separation, considered to be the result of shock induced boundary layer interactions.

8:39AM MX.00004 Linear Stability Analysis on Multiple Solutions of Steady Transonic Small Disturbance Equation , YA LIU\footnote{Graduated Student}, FENG LIU\footnote{Professor}, SHIJUN LUO\footnote{Researcher}, Department of Mechanical and Aerospace Engineering, University of California, Irvine, CA 92697-3975 — Multiple solutions of the steady Transonic Small Disturbance (TSD) potential equation occur within a narrow range of freestream Mach number. Three numerical solutions for the NACA 0012 airfoil at freestream Mach number 0.85 and zero angle of attack are computed using the conservative Murman-Cole scheme. One solution is symmetric and the other two are asymmetric and mirror-images of each other. The linear stability of the numerical solutions are analyzed by an eigenvalue technique. The Jacobi matrix is constructed from the discrete steady TSD equation and evaluated with the converged numerical solution. The maximum real part of the eigenvalues for the symmetric solution is positive $(0.7 \times 10^{-2})$ and that for the asymmetric solution is negative $(-0.8 \times 10^{-2})$ indicating that the symmetric solution is unstable and the two asymmetric solutions are stable under small perturbations. These stability conclusions are verified by numerical computations.

8:52AM MX.00005 Numerical flow simulation of a reusable sounding rocket during nose-up rotation , KAZUTO KUZU, KEICHI KITAMURA, KEICHIRO FUJIMOTO, EIJJI SHIMA, Researcher — Flow around a reusable sounding rocket during nose-up rotation is simulated using unstructured compressible CFD code. While a reusable sounding rocket is expected to reduce the cost of the flight management, it is demanded that this rocket has good performance for wide range of flight conditions from vertical take-off to vertical landing. A rotating body, which corresponds to a vehicle’s motion just before vertical landing, is a flight environment that largely affects its aerodynamics. Unlike landing of the space shuttle, this vehicle must rotate from gliding position to vertical landing position in nose-up direction. During this rotation, the vehicle generates massive separations in the wake. As a result, induced flow becomes unsteady and could have influence on aerodynamic characteristics of the vehicle. In this study, we focus on the analysis of such dynamic characteristics of the rotating vehicle. An employed numerical code is based on a cell-centered finite volume compressible flow solver applied to a moving grid system. The moving grid is introduced for the analysis of rotating motion. Furthermore, in order to estimate an unsteady turbulence, we employed DDES method as a turbulence model. In this simulation, flight velocity is subsonic. Through this simulation, we discuss the effect on aerodynamic characteristics of a vehicle’s shape and motion.
9:05AM MX.00006 Qualitative CFD for Rapid Learning in Industrial and Academic Applications

Evan Variano, University of California, Berkeley — We present a set of tools that allow CFD to be used at an early stage in the design process. Users can rapidly explore the qualitative aspects of fluid flow using real-time simulations that react immediately to design changes. This can guide the design process by fostering an intuitive understanding of fluid dynamics at the prototyping stage. We use an extremely stable Navier-Stokes solver that is available commercially (and free to academic users) plus a custom user interface. The code is designed for the animation and gaming industry, and we exploit the powerful graphical display capabilities to develop a unique human-machine interface. This interface allows the user to efficiently explore the flow in 3D + real time, fostering an intuitive understanding of steady and unsteady flow patterns. There are obvious extensions to use in an academic setting. The trade-offs between accuracy and speed will be discussed in the context of CFD’s role in design and education.

Thanks to financial and technical support from Autodesk Inc. and the Autodesk IDEAStudio.

9:18AM MX.00007 Characterization of Synthetic GTL Jet Fuel for use in Gas Turbine Engines

Reza Sadr, Kumaran Kannayain, Texas A&M at Qatar, Rolls Royce, UK Team, DLR, Germany Team, Shell, Qatar Team — Stringent emission regulations have instigated the search for alternative-clean source of energy. Recently, Gas-to-Liquid (GTL) fuel has grabbed the global attention by its clean combustion characteristics owing to the absence of aromatics and Sulphur. However, this will introduce potential risks and benefits. Last fall Qatar airways has proven the feasibility of using GTL as a potential alternative clean fuel by a 3200 mile flight using a fuel blend of 50% Jet A + 50% GTL. Researchers from Texas A & M University at Qatar (TAMUQ) in collaboration with their counterparts in Rolls-Royce (RR), UK, and German Aerospace Laboratory (DLR) are in a joint effort to establish an in-depth characterization of the combustion performance of GTL fuel in gas turbine engines. In TAMUQ, the research focus is to investigate the spray characteristics of GTL fuels. The results will be compared with that of standard fuel and correlate with combustion results to gain insights on GTL performance. This will help designers to optimize the nozzle geometry to improve the combustor performance. The objective of this talk is to introduce this ongoing effort and to discuss the experimental facility and preliminary results.

9:31AM MX.00008 Coaxial twin-fluid atomization with pattern air gas streams

Alberto Aliseda, University of Washington — Coaxial twin-fluid atomization has numerous industrial applications, most notably fuel injection and spray coating. In the coating process of pharmaceutical tablets, the coaxial atomizing air stream is accompanied by two diametrically opposed side jets that impinge on the liquid/gas coaxial jets at an angle to produce an elliptical shape of the spray’s cross section. Our study focuses on the influence of these side jets on the break up process and on the droplet velocity and diameter distribution along the cross section. The ultimate goal is to predict the size distribution and volume flux per unit area in the spray. With this predictive model, an optimal atomizing air/pattern air ratio can be found to achieve the desired coating result. This model is also crucial in scaling up the laboratory setup to production level. We have performed experiments with different atomized liquids, such as water and glycerine-water mixtures, that allow us to establish the effect of liquid viscosity, through the Ohnesorge number, in the spray characteristics. The gas Reynolds number of our experiments ranges from 9000 to 18000 and the Weber number ranges from 400 to 1600. We will present the effect of pattern air in terms of the resulting droplets size, droplet number density and velocity at various distances downstream of the nozzle where the effect of pattern air is significant.

9:44AM MX.00009 Large eddy simulation on a pulverized coal combustion furnace with a complex swirl burner

Hiroaki Watanabe, Kenji Tanno, Criepi, Ryoichi Kurose, Satoru Komori, Kyoto University — Large-eddy simulation (LES) is applied to a pulverized coal combustion field in a combustion test furnace with a complex swirl burner called the advanced low NOx burner CI-alpha, and its validity is investigated by comparing with the experiment. The motion of coal particles is calculated by the Lagrangian method with a parcel model. In the coal combusting modeling, three chemical processes are considered, namely devolatilization, char combustion and gaseous reactions. The direct closure SSSR (scale similarity filtered reaction rate model) is employed as a turbulent combustion model. The results show that a swirling recirculation flow is formed in a central region close to the burner and its size and strength dynamically change with time. The predicted distributions of time-averaged variance of particle velocity and time-averaged gaseous temperature, oxygen and NO concentrations are in general agreement with the experiment.

9:57AM MX.00010 Hydrophobic coating study for anti-icing aircraft

Katsuaki Morita, University of Tokyo, Akihito Aoki, Akihisa Konno, Kogakuin University, Hirotaka Sakaue, JAXA — Anti-icing or deicing of an aircraft is necessary for a safe flight operation. Mechanical processes, such as heating and deicer boot, are widely used. Deicing fluids, such as ethylene glycol type, are used to coat the aircraft. However, these should be coated every time before the take-off, since the fluids come off from the aircraft while cruising. We study a hydrophobic coating as a anti-icing for an aircraft. It is designed to coat the aircraft without removal. Since a hydrophobic coating prevents water by reducing the surface energy, it would be another way to prevent ice on the aircraft. We provide a temperature-controlled room, which can control its temperature at an icing condition (-10 to 0 degrees C). The contact angle is tested for various hydrophobic coatings. A water jet impingement on a hydrophobic-coated plate is included. The jet freezes under the icing condition. Qualitative comparison among various hydrophobic coatings as anti-icing is discussed.

Tuesday, November 23, 2010 8:00AM - 10:10AM
Session MY Magnetohydrodynamics
Hyatt Regency Long Beach Regency E

8:00AM MY.00001 Instabilities of conducting fluid flows in cylindrical shells under external forcing

Javier Burguete, Montserrat Miranda, University of Navarra — Flows created in neutral conducting flows remain one of the less studied topics of fluid dynamics, in spite of their relevance both in fundamental research (dynamo action, turbulence suppression) and applications (continuous casting, aluminium production, biophysics). Here we present the effect of a time-dependent magnetic field parallel to the axis of circular cavities. Due to the Lenz’s law, the time-dependent magnetic field generates an azimuthal current, that produces a radial force. This force produces the destabilization of the static fluid layer, and a flow is created. The geometry of the experimental cell is a disc layer with external diameter smaller than 94 mm, with or without internal hole. The layer is up to 20mm depth, and we use as conducting fluid an In-Ga-Sn alloy. There is no external current applied on the problem, only an external magnetic field. This field evolves harmonically with a frequency up to 10Hz, small enough to not to observe skin depth effects. The magnitude ranges from 0 to 0.1 T. With a threshold of 0.01T a dynamical behaviour is observed, and the main characteristics of this flow have been determined: different temporal resonances and spatial patterns with different symmetries (squares, hexagonal, triangles,...).

Financial support: FIS2008-01126 Spanish Gov.

1Supported by QSTP.
8:13AM MY.00002 An all-speed projection method for magnetohydrodynamics1, RAVI SAMTANEY, King Abdullah University of Science and Technology, MARK ADAMS, Columbia University, PHILLIP COLELLA, DANIEL GRAVES, TERRY LIGOCKI, BRIAN VAN STRAALEN, Lawrence Berkeley National Laboratory — We present an all-speed algorithm for magnetohydrodynamics (MHD) similar to the work of Colella & Pao (J. Comput. Phys. 1999) for low speed hydrodynamics. The method is based on an asymptotic ordering of scales relevant for tokamak MHD physics. The central idea is to Hodge decompose the velocity, and a splitting of the magnetic guide field analogous to the pressure splitting in low Mach number hydrodynamics into a thermodynamic and an incompressible part. We present the derivation of the ideal MHD equations into slow (advective), intermediate and fast scales. The algorithm treats the slow advective scales using a Godunov-procedure while the intermediate and fast scales are treated implicitly using backward Euler. Results from numerical tests such as the MHD Kelvin-Helmholtz instability, magnetic reconnection and others will be presented. We will highlight the challenges of designing solvers for the intermediate scales.

8:26AM MY.00003 Direct Numerical Simulation for the MHD Homogeneous Shear Turbulence with the Uniform Magnetic Field, MASAYOSHI OKAMOTO, Department of the Mechanical Engineering, Shizuoka University — The MHD homogeneous shear turbulent flows with the streamwise or span uniform magnetic field are investigated by means of the direct numerical simulation. The kinetic and magnetic energy in the streamwise uniform magnetic field cases are slightly small in comparison with the zero mean magnetic field case. The energy transformation term between the kinetic and magnetic energy, which is related with only fluctuating-field quantities, is balanced with that of the mean magnetic field. Under the spanwise magnetic field, the kinetic and magnetic energy is strongly suppressed and the Reynolds and Maxwell stresses become isotropic. The negative production term of the kinetic energy plays an important role in the energy suppression. The contribution of the energy transformation term of the fluctuating field is small unlike the streamwise magnetic field cases. From the energy-spectral viewpoints, the additional spanwise magnetic field has an influence on the large-scale factors.

8:39AM MY.00004 Transition in MHD duct flow, OLEG ZIKANOV, University of Michigan - Dearborn, DMITRI KRASNOV, Ilmenau University of Technology, MAURICE ROSSI, Universite Pierre et Marie Curie, THOMAS BOECK, Ilmenau University of Technology — A magnetic field applied to a flow of an electrically conducting fluid (e.g., a liquid metal) suppresses turbulence and can transform the flow into a weakly turbulent or laminar state. This situation is common in technical applications, such as metallurgy, materials processing, and liquid metal cooling for fusion reactors. The understanding and prediction of transition in MHD flows is therefore not only of interest from a theoretical perspective. We investigate the transition in the flow in a rectangular duct with electrically insulating walls and uniform transverse magnetic field. The essential features of the flow are the sidewall and Hartmann boundary layers on the walls parallel and perpendicular to the applied field. The transition mechanism based on the transient finite-amplitude growth is analyzed using numerical simulations with a highly conservative finite-difference scheme. The presented results include the identification and parametric study of optimal modes, which are found to be three-dimensional and localized in the sidewall layers. We also investigate, using the DNS approach, the growth and breakdown of the optimal modes leading to turbulence. Financial support is by the DFG (Bo 1668/2-4, 1668/5-1) and NSF (CBET 096557).

8:52AM MY.00005 Direct numerical simulations of turbulent MHD flow in a 2:1 aspect ratio rectangular duct subjected to transverse and span-wise magnetic fields, R. CHAUDHARY, A.F. SHINN, B.G. THOMAS, S.P. VANKA, University of Illinois at Urbana-Champaign — Magnetic fields are used to control flows in a variety of applications, notably in materials processing. One such process is continuous casting of steel which uses different types of magnetic fields to alter the flow, inclusion transport and multiphase flow in order to reduce defects in cast steel. Under a strong magnetic field, a turbulent flow can be altered significantly to the point that turbulence is completely suppressed and the flow is laminarized. In the present study, we have considered a periodic duct with an aspect ratio of 2:1 and subjected to a magnetic field either on the broad side or on the narrower side. We have conducted DNS of five different cases, with two magnetic field intensities in either direction and compared them with the case of no magnetic field. Calculations have been performed for Hartmann numbers of 0, 6.0 and 8.5 at a Reynolds number (based on bulk velocity) of 5000. The grid used consisted of 512 x 120 x 224 control volumes with grid stretching in the cross-stream directions. Various turbulence and mean flow statistics have been computed to characterize the effects of the magnetic field. We believe these results can be valuable additions to the databases that can be used for turbulence model development.

9:05AM MY.00006 Numerical simulation of quasi-static magnetohydrodynamic flow in a right-angle bend, STIJN VANTIEGHEM, BERNARD KNAEPEN, Universite Libre de Bruxelles, PHYSIQUE STATISTIQUE ET DES PLASMAS TEAM — We discuss simulation results of magnetohydrodynamic flows in a right-angle bend in the limit of vanishing magnetic Reynolds number. Both the Hartmann number and the magnetic parameter are much larger than one (these are non-dimensional estimates of the electromagnetic interaction with respect to the viscous term, respectively to the inertial term). Such a configuration is of interest in the context of the design of self-cooled blankets for future fusion devices. We consider the situation in which the magnetic field is perfectly aligned with the outflow direction, as well as a so-called backward elbow, in which the magnetic field lines make a positive angle with the to the outflow direction. Both cases are characterized by a thin, intense shear layer, aligned with the magnetic field, in the vicinity of the inner corner. We present results for steady as well as non-steady cases and various values of the Hartmann number. These results will be compared to existing asymptotic analysis and experimental data.

9:18AM MY.00007 Gyroviscous effects in channel flow with Braginskii magnetohydrodynamics, PAUL DELLAR, University of Oxford — We study the pressure-driven flow across a magnetic field of a fluid obeying magnetohydrodynamics with the full Braginskii stress tensor. Momentum transport across magnetic field lines is strongly suppressed by the spiraling of charged particles around the field lines. This spiraling also creates gyroviscous stresses perpendicular to both the strain rate and the magnetic field. These stresses are negligible in the bulk of the flow, but they alleviate the singularities in current and shear that otherwise occur at the channel walls. They create boundary layers whose width scales as the three-quarters power of the ratio $\epsilon = \mu_\parallel /\mu_\perp$ of the gyro to parallel viscosities. The maximum current and velocity scale as $\epsilon^{-1/4}$. The gyroviscous stresses generate a leading-order flow perpendicular to the plane defined by the pressure gradient and imposed magnetic field, so the maximum velocity is tilted close to 40° out of this plane. The related out of plane magnetic field scales as $\epsilon^{1/2}$ in the boundary layers, and as $\epsilon$ elsewhere. Regularisation by gyroviscous stresses alone is sufficient. Perpendicular viscosity makes only a further local perturbation.

9:31AM MY.00008 Measuring and analyzing the magnetic field in (SERAJ system) theta pinch device using the Magnetic probes, HAMIDA ESHRAEE, EHSAN ELSHUMMAKHI, Al-Fateh University, Tripoli-Libya, Plasma Research Laboratory — Using the internal and the external magnetic probes in different positions inside and outside plasma discharge chamber of Seraj's Theta-pinch system (L = 150cm, D = 8.4cm), the generated magnetic field on the coil of Seraj's system has been calculated. When the plasma is discharged (by discharging the four capacitors bank connected on parallel, the capacity of each is (12.5uF) in the system coil (12nH)) the characteristics of plasma can be defined as how much magnetic field is affected. Comparing the magnetic field in the absence or presence of plasma, trapped magnetic field and Damagnetic effect could be determined in the Theta pinch system. In the present study, one could determine the appropriate operating circumstances to produce suitable plasma with specific features (density & electron temperature) as an advantage for different application.
The dynamics of mercury flow in a curved pipe, YAN ZHAN, FOLUSO LADEINDE, Dept. of Mechanical Engineering, Stony Brook University — Mercury has been investigated as a potential high-Z target for the Moun Collider Accelerator project. Preliminary design of the target delivery system involves pipe curvature and axially-dependent pipe radius. The investigation of the dynamics of mercury flow under these conditions is undertaken with the goal of obtaining the proper nozzle design for this application. Depending on the Dean number, rotational body force modes are observed, with dynamics that are considerably different from those in a straight pipe of constant radius.
8:52AM MZ.00005 Influence of the surface tension and the viscosity on irregular surface switching of rotating fluids, YUJI TASAKA, MAKOTO IIMA, Hokkaido University — We have investigated the influence of the surface tension and the viscosity of the fluids on temporally-irregular surface switching, which is a recently discovered phenomenon of rotating fluids accompanied by a free surface deformation (see Suzuki et al., Phys. Fluids, 2006 and Tasaka & Iima, J. Fluid Mech., 2009). Tap water, liquid Gallium, 1 cSt and 10 cSt Silicone oils were used to change the viscosity and the surface tension; oils have smaller surface tension than water and liquid gallium larger one than water. Variations of the magnitude of the velocity fluctuation with respect to the speed of the disk rotation were obtained for the fluids without liquid gallium as the bifurcation diagram. Comparison of the diagram between the fluids indicates that the smaller surface tension enhances the surface deformation but prevents the appearance of the surface switching. Surface motion recorded by a digital video camera shows that the surface of the rotating oil(1 cSt) has sloshing motion ($n = 1$) ahead of the surface switching instead of the intermittent deformation of the surface shape ($n = 2$) which was observed in water; here $n$ means the mode of axisymmetry breaking on the free surface.

9:05AM MZ.00006 Studies of Rossby waves and hydrodynamic turbulence in a Taylor-Couette device, ERIC EDLUND, Princeton Plasma Physics Laboratory, E. SCHARTMAN, Nova Photonics, E. SPENCE, A. ROACH, P. SLOBODA, H. JI, Princeton Plasma Physics Laboratory — We present the design of a new experiment at the Princeton Plasma Physics Laboratory with the mission of studying angular momentum transport in rotating incompressible fluids at Re $> 10^6$. This hydrodynamic experiment supports and complements a similar device, the Princeton MRI experiment, which uses a liquid metal to study MHD effects [1]. The inner and outer cylinders may be separately driven; differentially rotating rings on the top and bottom boundaries between the cylinders allow the Ekman circulation to be greatly diminished while maintaining shear in the azimuthal flow close to the Rayleigh criterion. The top, fluid-facing boundary of the device can be outfitted with various surfaces or operated with a free surface to modify the Rossby wave characteristics. A set of ultrasonic transducers is used to measure the $v_\phi$ and $v_r$ profiles at three distinct heights. A two component LDV system provides measurements of the local $v_r$ and $v_\phi$ which will further constrain measurements of the turbulent angular momentum transport reported previously [2].


9:18AM MZ.00007 A Stereo-PIV Study of the Taylor-Column Generated by a Rotating Disk, JOZEF H.A. VLASKAMP, PETER J. THOMAS, University of Warwick, RAINER HOLLERBACH, University of Leeds, ROBERT M. KERR, University of Warwick — The formation of Taylor-Coulombs is one of the familiar phenomena observed in flows where strong background rotation is present. The current investigation considers the Taylor-Column generated by a rotating disc in a rotating fluid, a geometry similar to the classic Stewartson-layer problem. Experimental work is performed on the large turntable at the University of Warwick (overall height 5.7m and 1.4m diameter). The facility offers a water depth of 2m below the disk, allowing for a much longer Taylor-Coulomb to be observed than in previous experimental work. A fully automated, traverse-mounted Stereo-PIV system has been developed to visualize the flow, allowing data-acquisition at different heights of the Taylor-Column without the need for recalibration. The experimental results show a z-dependence of the angular velocity profile in the Taylor-Column, which contradicts the Taylor-Proudman theorem. Additional numerical simulation is being performed and compared with the experimental results.

9:31AM MZ.00008 Velocity profile measurements in high Reynolds number Taylor-Couette flow for pure outer-cylinder rotation, SANDER HUISMAN, DENNIS VAN GILS, CHAO SUN, Physics of Fluids group, University of Twente, GERTJAN VAN HEIJST, Turbulence and Vortex Dynamics, Technical University of Eindhoven, DETLEF LOHSE, Physics of Fluids group, University of Twente — Using Laser Doppler Anemometry, we measured azimuthal and axial velocity profiles inside the gap of a Taylor-Couette apparatus, spinning only the outer cylinder at a varying Reynolds number from $1.4 \times 10^5$ to $1.4 \times 10^6$. The system has a radius ratio of 0.716 and an aspect ratio of 11.68, and the end plates are attached to the outer cylinder. We found the laminar velocity profile due to the influence of the end plates. We analyzed the end effects by studying the Ekman and Stewartson boundary layer dynamics in Taylor-Couette flow.

9:44AM MZ.00009 Statistics of velocity fluctuations in turbulent Couette-Taylor flow, CHAO SUN, SANDER HUISMAN, DENNIS VAN GILS, DETLEF LOHSE, Physics of Fluids group, University of Twente — Using Laser Doppler Anemometry in a turbulent Taylor-Couette apparatus, we measured the velocity fluctuations in the middle of the gap and near the outer wall, for pure inner cylinder and counter-rotating cylinders at a fixed Reynolds number of $2 \times 10^5$. The velocity structure functions calculated using extended self-similarity exhibit clear power-law scaling. The transverse and longitudinal structure function exponents have been compared in two measurement positions.

9:57AM MZ.00010 Torque scaling between independently rotating cylinders in turbulent Taylor-Couette flow, DENNIS VAN GILS, DANIILA NAREZO, SANDER HUISMAN, CHAO SUN, DETLEF LOHSE, University of Twente — The Twente Turbulent Taylor-Couette (T²C) system allows for a 20 Hz rotation rate of the inner cylinder and 10 Hz rotation rate of the outer cylinder. The corresponding maximum Reynolds numbers for the inner and outer cylinder are Re$_r$ = 2.0 $\times 10^5$, and Re$_o$ = 1.4 $\times 10^6$, respectively. The system has a radius ratio of 0.716 and an aspect ratio of 11.68, and the end plates are attached to the outer cylinder. We measured the global torque (G) as a function of the inner and outer cylinder Reynolds numbers in the unexplored parameter space of co- and counter-rotation.

Tuesday, November 23, 2010 10:30AM - 11:05AM — Session NS Invited Session: Turbulence in Strongly Stratified Flows Long Beach Convention Center Grand Ballroom A

10:30AM NS.00001 Turbulence in Strongly Stratified Flows1, JAMES RILEY, University of Washington — Laboratory experiments in the early 1970’s (e.g., Lin & Pao, Ann. Rev. Fluid Mech., 11, 1979) revealed quasi-horizontal vortices in wakes of objects moving through stably-stratified fluids. These results suggested new flow dynamics, in addition to internal waves, in flows strongly affected by stable density stratification, conditions which often occur, e.g., in the atmosphere and oceans. In the past ten years theoretical analyses, numerical simulations, laboratory experiments and some field studies have led to a much better understanding of these flows and, in particular, how they can provide pathways to three-dimensional turbulence. In this talk some of the results of this research will be discussed.

1 ARO Grant No. W911NF-08-1-0155; NSF Grant No. OCI-0749209

Tuesday, November 23, 2010 10:30AM - 11:05AM — Session NT Invited Session: Wetting, Spreading and Capillary Adhesion: Putting Shape-instability to Purpose Long Beach Convention Center Grand Ballroom B
allow one to compute both the flow inside interfacial droplets and the flow in the layer of liquid substrate supporting the droplet and the lessons which can be
in the past. In contrast, the motion of droplets confined to the free surface of a liquid substrate has received much less attention. Recent developments in
tangential surface stresses arising from the variation of surface tension with temperature create a propulsive force on the bubble–has been extensively studied
drag reduction by superhydrophobic surfaces in turbulent channel flow
control the contact angle at the interfaces, simulating hydrophobic or superhydrophobic on the solid walls. Test results in channel flow will be presented
AKHAVAN, University of Michigan — A lattice Boltzmann code for direct numerical simulation of flow over superhydrophobic surfaces has been developed.
droplets
[1] Very peculiar nature of quantum turbulence. One of many surprising observations is the existence of power-law tails in the probability distribution of velocities
have directly confirmed the two-fluid model, observed vortex rings and quantized vortex reconnection, characterized thermal counterflows, and observed the
tangle of quantized vortices, as first envisioned by Feynman. Approximately five years ago, our group discovered that micron-sized hydrogen particles may be
used for flow visualization in superfluid helium-4. The particles can trace the motions of the normal fluid or be trapped by the quantized vortices, which enables one
to characterize the dynamics of both the normal fluid and superfluid components for the first time. By directly observing and tracking these particles, we
have directly confirmed the two-fluid model, observed vortex rings and quantized vortex reconnection, characterized thermal counterflows, and observed the
very peculiar nature of quantum turbulence. One of many surprising observations is the existence of power-law tails in the probability distribution of velocities
in quantum turbulence, which are in stark contrast to the Gaussian distributions typical of classical fluid turbulence.

Tuesday, November 23, 2010 11:10AM - 11:30AM –
Session PS Andreas Acrivos Dissertation Prize Lecture: Quantum Mechanics meets
Fluid Dynamics: Visualization of Vortex Reconnection in Superfluid Helium , MATTHEW PAOLETTI
University of Maryland and University of Texas at Austin — Long-range quantum order underlies a number of related physical phenomena including superfluidity,
superconductivity and Bose-Einstein condensation. While superfluidity in helium-4 was one of the earliest discovered, it is not the best understood, owing to
the strong interactions present (making theoretical progress difficult) and the lack of local experimental probes. Quantum fluids, such as superfluid helium-4,
are typically described as a mixture of two interpenetrating fluids with distinct velocity fields: a viscous normal fluid akin to water and an inviscid superfluid
exhibiting long-range quantum order. In this “two-fluid model,” there is no conventional viscous dissipation in the superfluid component and vorticity is confined
to atomically-thin vortices with quantized circulation. Turbulence may occur in either fluid component with turbulence in the superfluid exhibiting a complex
tangle of quantized vortices, as first envisioned by Feynman. Approximately five years ago, our group discovered that micron-sized hydrogen particles may be
used for flow visualization in superfluid helium-4. The particles can trace the motions of the normal fluid or be trapped by the quantized vortices, which enables one
to characterize the dynamics of both the normal fluid and superfluid components for the first time. By directly observing and tracking these particles, we
have directly confirmed the two-fluid model, observed vortex rings and quantized vortex reconnection, characterized thermal counterflows, and observed the
very peculiar nature of quantum turbulence. One of many surprising observations is the existence of power-law tails in the probability distribution of velocities
in quantum turbulence, which are in stark contrast to the Gaussian distributions typical of classical fluid turbulence.

Tuesday, November 23, 2010 11:10AM - 11:30AM –
Session PT Francois Frenkiel Award Lecture Long Beach Convention Center Grand Ballroom B

11:10AM PS.00001 Francois Frenkiel Award Lecture: Thermocapillary migration of interfacial
droplets , EDWIN F. GRECO1, Georgia Institute of Technology — Thermocapillary migration of bubbles through the bulk liquid—a process in which
tangential surface stresses arising from the variation of surface tension with temperature create a propulsive force on the bubble—has been extensively studied
in the past. In contrast, the motion of droplets confined to the free surface of a liquid substrate has received much less attention. Recent developments in
microfluidics provided new motivation to understand how applied thermal gradients can affect the motion of, and mixing inside, small aqueous droplets. In
particular, the quality and speed of mixing depend rather sensitively on the flow structure inside the droplet. In this talk we describe different approaches that
allow one to compute both the flow inside interfacial droplets and the flow in the layer of liquid substrate supporting the droplet and the lessons which can be
learned by analyzing these flows.

Tuesday, November 23, 2010 12:50PM - 3:00PM – Session QA CFD: Multiphase and Mixing Applications Long Beach Convention Center 101A

12:50PM QA.00001 A lattice Boltzmann code for direct numerical simulation of skin-friction
drag reduction by superhydrophobic surfaces in turbulent channel flow , AMIRREZA RASTEGARI, RAYHANEH
AKHAVAN, University of Michigan — A lattice Boltzmann code for direct numerical simulation of flow over superhydrophobic surfaces has been developed.
The code solves the Boltzmann equation for two different sets of particle distribution functions based on the Shan and Chen model [1], to account for the
gas-liquid interactions. The immiscibility and inter-phase interactions are controlled through an interaction body force between the distribution functions. The
recently proposed model of Hunag et al. [2] is used to set the contact angle in the simulations, in which by tuning the values of the interaction force, one can
control the contact angle at the interfaces, simulating hydrophobicity or superhydrophobicity on the solid walls. Test results in channel flow will be presented
and discussed.


1:03PM QA.00002 Evaluation of CFD predictions of mixing properties of hydrogen as a fuel
in a model premixer by experimental and statistical tools , AMIN AKBARI, VINCE MCDONELL, SCOTT SAMUELSEN,
UCI, ADVANCED POWER AND ENERGY PROGRAM (APEP) TEAM — Mixing properties of hydrogen, as a new fuel of interest in combustion community
research, have been numerically studied utilizing various CFD approaches in both axial and radial fuel injection type premixer configurations to be applied in lean
combustion techniques research for design purposes. Numerical predictions of RANS and LES turbulent models have been evaluated by conducting experimental
measurements. Comprehensive qualitative and quantitative comparisons have been made between numerical and experimental results to investigate capabilities
of different CFD approaches to provide reliable predictions of flow field properties. Moreover, extensive statistical study has been accomplished, and ANOVA
results are interpreted beside other comparison approaches to draw fruitful conclusions. In general, sensitivity of numerical predictions to different turbulent
models as well as sensitivity to different turbulent Schmidt numbers has been explored. The result of comparison suggests more sensitivity to turbulent models
for radial injection configurations. However, more sensitivity to ScFl has been witnessed for the axial configuration. In general, RSM turbulent model with
ScFl=0.7 provides the most promising predictions for various combinations of different fuels and injection types.
1:16PM QA.00003 A Quadrature Free Discontinuous Galerkin Conservative Level Set Scheme

MARK CZAJKOWSKI, OLIVIER DESJARDINS, University of Colorado at Boulder — In an effort to improve the scalability and accuracy of the Accurate Conservative Level Set (ACLS) scheme (Desjardins et al., J COMPUT PHYS 227 (2008)), a scheme based on the quadrature free discontinuous Galerkin (DG) methodology has been developed. ACLS relies on a hyperbolic tangent level set function that is transported and reinitialized using conservative schemes in order to alleviate mass conservation issues known to plague level set methods. DG allows for an arbitrarily high order representation of the interface by using a basis of high order polynomials while only using data from the faces of neighboring cells. The small stencil allows DG to have excellent parallel scalability. The diffusion term present in the conservative reinitialization equation is handled using local DG method (Cockburn et al., SIAM J NUMER ANAL 39, (2001)) while the normals are computed from a limited form of the level set function in order to avoid spurious oscillations. The resulting scheme is shown to be both robust, accurate, and highly scalable, making it a method of choice for large-scale simulations of multiphase flows with complex interfacial topology.

1:29PM QA.00004 A Diffuse Interface Method with Adaptive Mesh Refinement for Simulation of Incompressible Multi-Phase Flows with Moving Contact Lines

YI SUI, PETER D.M. SPELT, Department of Chemical Engineering, Imperial College London, HANG DING, Department of Chemical Engineering, University of California, Santa Barbara — Diffuse Interface (DI) methods are employed widely for the numerical simulation of two-phase flows, even with moving contact lines. In a DI method, the interface thickness should be as thin as possible to simulate spreading phenomena under realistic flow conditions, so a fine grid is required, beyond the reach of current methods that employ a uniform grid. Here we have integrated a DI method based on a uniform mesh, to a block-based adaptive mesh refinement method, so that only the regions near the interface are resolved by a fine mesh. The performance of the present method is tested by simulations including drop deformation in shear flow, Rayleigh-Taylor instability and drop spreading on a flat surface, et al. The results show that the present method can give accurate results with much smaller computational cost, compared to the original DI method based on a uniform mesh. Based on the present method, simulation of drop spreading is carried out with Cahn number of 0.001 and the contact line region is well resolved. The flow field near the contact line, the contact line speed as well as the apparent contact angle are investigated in detail and compared with previous analytical work.

1:42PM QA.00005 Droplet deformation in shear flow: a comparison between multicomponent Lattice-Boltzmann method (LBM) and Phase Field method (PFM)

ALFREDO SOLDATI, LUCA SCARBOLO, DAFNE MOLIN, Università degli Studi di Udine, PRASAD PERLEKAR, FEDERICO TOSCHI, Technische Universität Eindhoven — Prediction of droplet breakup, droplet coalescence and phase separation are crucial in many industrial and environmental processes. We present a direct comparison between two numerical approaches on the problem of deformation and breakup of a droplet under shear flow conditions. In order to quantitatively benchmark the Lattice Boltzmann and the Phase Field methods we consider the same flow conditions. Through the comparison of the two approaches we can assess respective advantages and disadvantages. For different values of the dimensionless problem parameters we investigate quantitatively both physical properties as well as numerical issues related to the two different methodologies.

1:55PM QA.00006 Fluctuating Lattice Boltzmann

GOETZ, KAEHLER, ALEXANDER WAGNER, North Dakota State University — Fluctuations are important for many hydrodynamic phenomena such as colloid diffusion or phase-separation near the critical point. However, the standard LB-schemes do not take fluctuations into account. Based on Ladd’s [1] and later Adhikari’s [2] work we explain how one can practically implement noise in Lattice Boltzmann algorithms for different dimensionality and base velocity sets. We also present a new velocity moment method for deriving the hydrodynamic equations from the underlying Multi Relaxation Time models [3,4]. This sets the foundation for formulating more general fluctuating lattice Boltzmann methods for energy conserving thermal systems and multi-component systems.


2:08PM QA.00007 Lattice Boltzmann modeling for energy conversion systems

NIKOLAOS PRASIANAKIS, JOHN MANTZARAS, Paul Scherrer Institut — The study of advanced small energy conversion systems, such as fuel cells (SOFCs, PEFCs) and microcombustors, requires the use of lattice Boltzmann models that can handle heat transfer and mixing effects. Heat transfer effects can be efficiently studied on the standard lattices (D2Q9, D3Q7) with the thermal model introduced in Ref. [1]. This model can simulate compressible Navier-Stokes and Fourier equations for large temperature and density variations [2,3]. For multi-component isothermal flows, the model of Ref.[4] is used to simulate the flow through a porous anode of a SOFC. Simulation results show that by decreasing the characteristic length scale of the simulated geometry, micro flow effects that alter the flow field start to emerge. A model that combines the properties and the novelties of both aforementioned thermal and multi-component models will be outlined. References [1] N.I. Prasianakis, I.V. Karlin, Phys. Rev. E 76, 016702 (2006) [2] N.I. Prasianakis, PhD Thesis No 17739, ETH Zurich (2008) [3] N.I. Prasianakis, I.V. Karlin, J. Mantzaras, K. Boulouchos, Phys. Rev. E 79, 066702 (2009) [4] S. Arcidiacono, I.V. Karlin, J. Mantzaras, C.E. Frouzakis, Phys. Rev. E 76, 046703 (2007)

2:21PM QA.00008 Performance Limitation by Reactant Crossover in a Membraneless Fuel Cell

ISAAC SPRAGUE, PRASHANTA DUTTA, Washington State University — In the past decade, the paradigm of using micro fuel cells for portable power applications has inspired novel innovations in fuel cell technology. One such example is the laminar flow fuel cell (LFFC) which utilizes columnar flow to maintain the separation between the anode and cathode instead of a membrane. Although the membraneless LFFC provides a simple design for prototype development, the lack of a physical separation can permit oxidant as well as fuel to crossover and impact device performance adversely. To understand the effect of reactant crossover and its effect on the performance of LFFC a mathematical model is developed. The model includes a more general treatment of reactant crossover than the common method where it is assumed that the crossover flux is fully utilized as crossover current. This model is used to study the performance of a LFFC operating with different electrode lengths and separations. Numerical results show that the reactant crossover, transport limitations, and Ohmic losses are the primary performance limitation factors. The fuel cells with shorter channel heights suffer from transport limitations at the longer electrode lengths even when reactant crossover is neglected.
2:34PM QA.00009 A study on improving numerical stability by applying filter operation to concentration flux for reverse simulation, SATOSHI ABE, graduate school, Univ. of Tokyo, SHINSUKE KATO, IIS, Univ. of Tokyo — A reverse simulation based on CFD is considered to be a resolution for identifying pollutant source, which is the solution of a transport equation in negative time advancing. The process is equivalent to positive time advancing with negative diffusion and convection. However, there is a numerical instability problem when solving the negative diffusion. Therefore, in this study, we propose a method to improve numerical stability by applying a low-pass filter to concentration flux in RANS analysis. The following equations are summary of the low-pass filter. By setting an appropriate filter weight ($\Delta$), the reverse simulation can be realized. $F(x)$ is the concentration flux in RANS analysis. $F(x) = \int_{-\infty}^{\infty} G(r) F(x-r) dr$, $G(r) = \sqrt{\frac{\pi}{2}} \exp\left(-\frac{r^2}{2}\right)$. $F(x) = F_{\infty}$. The aim of filter operation to concentration flux is to stabilize concentration balance in control volume. As the result, the following conclusion can be drawn. 1. Filter operation is a useful method for improving numerical stability. 2. The concentration center moves and the standard division of concentration shrink adequately in reverse simulation, which makes it easy to identify pollutant source. In the next stage, we will develop a method to identify the pollutant source stochastically.

3This research is financially supported by the ASahi glass foundation.

2:47PM QA.00010 Coarse Grid CFD for underresolved simulation, ANDREAS G. CLASS, MATHIAS O. VIELLIEBER, STEFFEN R. HIMMEL2, Karlsruhe Institute of Technology — CFD simulation of the complete reactor core of a nuclear power plant requires exceedingly huge computational resources so that this crude power approach has not been pursued yet. The traditional approach is 1D subchannel analysis employing calibrated transport models. Coarse grid CFD is an attractive alternative technique based on strongly under resolved CFD and the inviscid Euler equations. Obviously, using incoherent grids and coarse grids does not resolve all the physics requiring additional volumetric source terms modelling viscosity and other sub-grid effects. The source terms are implemented via correlations derived from fully resolved representative simulations which can be tabulated or computed on the fly. The technique is demonstrated for a Carnot diffuser and a wire-wrap fuel assembly [1].


3This work is supported by the EU with grant FP7 THINS.

4Currently Siemens AG

Tuesday, November 23, 2010 12:50PM - 3:00PM — Session QB Turbulent Boundary Layers IX Long Beach Convention Center 101B

12:50PM QB.00001 Scale-separation effects on the mechanisms of turbulent inertia, CALEB MORRILL-WINTER, University of New Hampshire, PATHTHAGE PRIYADARSHANA, Emerson Process Management, JOSEPH KLEWICKI, University of New Hampshire — The wall-normal gradients of the Reynolds stress and turbulent kinetic energy have direct connection to the transport mechanisms of turbulent boundary layers. Moreover, these gradients can be shown to arise from the correlation between specific velocity and vorticity components. Importantly, these correlations must remain non-zero at indefinitely high Reynolds numbers if turbulent transport is also to remain a non-negligible dynamical mechanism. Such considerations motivate the investigation of the relevant velocity vorticity products under the condition of increasing scale separation. In the boundary layer, this condition occurs with increasing $\Delta$ (at fixed Reynolds number), and more importantly, with increasing Reynolds number. In this study we continue to interrogate data from the SLTEST site in Utah’s west desert to explore the behavior of the relevant velocity and vorticity component interactions at high Reynolds number, while making use of existing well-resolved laboratory data to quantify distance from the wall effects. Pre-multiplied power spectra and the associated cospectra are interpreted in the context of the known momentum source and sink behaviors of the Reynolds stress gradient.

1:03PM QB.00002 Coherent enstrophy production and turbulent dissipation in two-dimensional turbulence with and without walls, ROMAIN NGUYEN VAN YEN, Ecole Normale Superieure, MARIE FARGE, Ecole Normale Superieure and CNRS, KAI SCHNEIDER, Universitè d’Aix-Marseille — In the fully-developed turbulent regime dissipation becomes independent on the molecular viscosity of the fluid for three-dimensional incompressible flows when Reynolds number becomes large enough. We inquire if incompressible two-dimensional turbulent flows may exhibit a similar behaviour in the vanishing viscosity limit. For this we examine the viscosity dependence of the solutions of two-dimensional Navier-Stokes equations in both periodic and wall-bounded domains, for Reynolds numbers varying from $10^4$ to $10^5$. The vorticity field is split into coherent and incoherent parts by applying the wavelet filter used for CVS [1]. We find that for Reynolds larger than $10^5$ the coherent enstrophy dissipation rate tends to become independent of Reynolds, while the total enstrophy dissipation rate decays to zero logarithmically with Reynolds. In the wall-bounded case, we observe an additional production of enstrophy at the wall. As a result, coherent enstrophy diverges when Reynolds tends to infinity, but its time derivative seems to remain bounded independently of Reynolds. [1] M. Farge, K. Schneider and N. Kevlahan, 1999. Non-Gaussianity and Coherent Vortex Simulation for two-dimensional turbulence using an adaptive orthogonal wavelet basis. Phys. Fluids., 11(8), 2187-2201.

1:16PM QB.00003 Markovian properties of velocity increments in a high Reynolds number turbulent boundary layer, MAREN FREDBO, Norwegian University of Science and Technology (NTNU), MURAT TUTKUN, Norwegian Defence Research Establishment (FFI), Kjeller, Norway — Statistics of velocity increments in a flat plate turbulent boundary layer are investigated using the theory of Markov processes (J. Fluid Mech., Vol. 433, pp. 383-409, 2001). The database analyzed here is a subset of data taken in the 20 m long wind tunnel of Laboratoire de Mécénique de Lille (LML) using a hot-wire rake of 143 single wire probes. The Reynolds number based on momentum thickness, $Re_\theta$, tested in this study was 19 100. The freestream velocity of the tunnel and the boundary layer thickness at the measurement location were 10 m s$^{-1}$ and 30 cm respectively. Our analysis on the increments of longitudinal velocities at different wall-normal positions show that the flow exhibits Markovian properties when the separation ($\Delta r$) between different scales is on the order of the Taylor microscale, $\lambda$. Initial results indicate that smallest $\Delta r/\lambda$, where the process can be defined as Markovian, decreases from wall to the inertial layer. As the probe moves inside the inertial layer, however, a constant $\Delta r/\lambda$ is observed. The ratio starts growing in the outer layer once the probe leaves the inertial layer.

3We acknowledge support from the Association CEA-Euratom in the framework of the European Fusion Development Agreement.

1:16PM QB.00003 Markovian properties of velocity increments in a high Reynolds number turbulent boundary layer, MAREN FREDBO, Norwegian University of Science and Technology (NTNU), MURAT TUTKUN, Norwegian Defence Research Establishment (FFI), Kjeller, Norway — Statistics of velocity increments in a flat plate turbulent boundary layer are investigated using the theory of Markov processes (J. Fluid Mech., Vol. 433, pp. 383-409, 2001). The database analyzed here is a subset of data taken in the 20 m long wind tunnel of Laboratoire de Mécénique de Lille (LML) using a hot-wire rake of 143 single wire probes. The Reynolds number based on momentum thickness, $Re_\theta$, tested in this study was 19 100. The freestream velocity of the tunnel and the boundary layer thickness at the measurement location were 10 m s$^{-1}$ and 30 cm respectively. Our analysis on the increments of longitudinal velocities at different wall-normal positions show that the flow exhibits Markovian properties when the separation ($\Delta r$) between different scales is on the order of the Taylor microscale, $\lambda$. Initial results indicate that smallest $\Delta r/\lambda$, where the process can be defined as Markovian, decreases from wall to the inertial layer. As the probe moves inside the inertial layer, however, a constant $\Delta r/\lambda$ is observed. The ratio starts growing in the outer layer once the probe leaves the inertial layer.

3This work has been performed under the WALLTURB project funded by the CEC under the 6th F.P. (Contract No:AST4-CT-2005-516008).
1:29PM QB.00004 Time-evolution and time-scales of topological structures in a turbulent boundary layer through conditional mean trajectory analysis1, CALLUM ATKINSON, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University, SERGEI CHUMAKOV, IVAN BERMEJO-MORENO, Center for Turbulence Research, Stanford University, XIAOHUA WU, Department of Mechanical Engineering, Royal Military College of Canada, JULIO SORIA, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University — The Lagrangian evolution of the invariants of the velocity gradient tensor in wall-bounded turbulence is examined using conditional mean trajectories (CMT). Fields from direct numerical simulations of a turbulent boundary layer developing over a flat plate with fully turbulent flow over a Reynolds number range of Re\textsubscript{θ} = 730 to 1954 are used to extract the CMT in the invariant space of the velocity gradient tensor \((Q_{ij}, R_{ij})\), the invariant space of the strain-rate tensor \((Q_{ij}, R_{ij})\) and the invariant space of the rate-of-rotation tensor \((Q_{ij}, R_{ij})\). CMT are considered for the full boundary layer, the log layer and the log and buffer layers. Results show a cyclic evolution of local topology. Associated time scales are extracted and compared with homogeneous isotropic turbulence.

1Supported by CTR Summer Program and the Australian Research Council.

1:42PM QB.00005 Geometrical structure and topology of pressure Hessian in the turbulent boundary layer1, SERGEI CHUMAKOV, Center for Turbulence Research, Stanford University, CALLUM ATKINSON, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University, IVAN BERMEJO-MORENO, Center for Turbulence Research, Stanford University, JULIO SORIA, Department of Mechanical and Aerospace Engineering, Monash University, Australia, XIAOHUA WU, Royal Military College of Canada — Pressure Hessian \(H_{ij} = F_{ij}\) plays a significant role in the evolution equations for the invariants of the deformation tensor \(A_{ij} = u_{i,j}\) and its symmetric part \(S_{ij}\). The properties of \(H_{ij}\) need to be understood in order to develop a mathematical model for the evolution of invariant quantities. In order to develop a full dynamical model for \(H_{ij}\), there is a need to study and understand the full effect of the \(H_{ij}\) tensor on the Lagrangian dynamics of the invariants. This type of study requires well-resolved data to evaluate all the right-hand side terms in the evolution equations. Attempts to study the properties of \(H_{ij}\) via its invariants for the case of decaying isotropic turbulence and a temporally evolving plane wake can be found in the current literature. We present a priori study of properties of \(H_{ij}\) based on the results from the DNS of the fully developed turbulent boundary layer over a smooth flat plate, originally performed by Wu and Moin.

1The work done as a part of biennial Summer School at Center for Turbulence Research at Stanford University.

1:55PM QB.00006 Identification of Lagrangian Coherent Structures in a Turbulent Boundary Layer 1, ZACHARY WILSON, Portland State University (PSU), MURAT TUTKUN, Norwegian Defence Research Establishment, RAUL BAYOAN CAL, PSU — In this study, we identify Lagrangian coherent structures (LCS) in a flat plate turbulent boundary layer at Re\textsubscript{θ} of 19,100. To detect the LCS, we compute direct Lyapunov exponents (DLE) (Haller, G., Physica D, vol 149, pp 248-277, 2001). Specifically we use the velocity field obtained from stereo PIV measurements to compute trajectories, \(x(t, t_0, x_0)\), from initial positions, \(x_0\), at time \(t_0\). For fixed integration times, \(|t-t_0|\), we numerically differentiate the flow map, given by \(F_{t_0}(x_0) = x(t, t_0, x_0)\), and then compute the deformation gradient tensor field \(\Delta_{ij}(x_0) = [\nabla F_{t_0}(x_0)]^T \cdot [\nabla F_{t_0}(x_0)]\). The DLE field is then found as \(\text{DLE}^i_{ij} = \ln \left| \frac{\Delta_{ij}(x_0)}{\Delta_{ij}(x_0)} \right| \big/ (2 |t-t_0|)\). Two dimensional gradient climbing is then used to find points on the locally maximizing, LCS surfaces of the field, \(\text{DLE}^i_{ij}\). To determine whether these surfaces truly repel (attract) nearby fluid particles, the hyperbolicity criterion is applied (Mathur et al., Phys. Rev. Lett., vol 98, pp 144502, 2007). In particular we compute normal strain rates, \(\langle n, S_n \rangle\), to locate repelling surfaces \((t >> t_0\) and \(\langle n, S_n \rangle > 0\)\) and attracting surfaces \((t << t_0\) and \(\langle n, S_n \rangle < 0\)\) within the boundary layer.

2:08PM QB.00007 The parametric mechanism maintaining the roll/streak/turbulence complex in boundary layers, BRIAN FARRELL, Harvard University — Stochastic Structural Stability Theory (SSST) provides an autonomous, deterministic, nonlinear dynamical system for evolving the statistical mean state of a turbulent system. In this presentation SSST is applied to the problem of understanding the maintenance of the roll/streak/turbulence complex that supports boundary layer turbulence. In the presence of sufficiently high levels of free stream turbulence roll/streak structures bifurcate from the laminar flow as a linear instability of interaction between the free stream turbulence and the mean flow leading to an essentially time dependent state that is self-maintaining in the absence of external forcing by free stream turbulence. This chaotic state is supported by the universal instability inherent in time dependent non-normal system dynamics.

2:21PM QB.00008 Classification of critical points in a turbulent boundary layer, MAX GIBSON, Portland State University (PSU), ANDERS HELGELAND, Norwegian Defence Research Establishment (FFI), MURAT TUTKUN, FFI, RAUL BAYOAN CAL, PSU — Critical points of the velocity field generated by the direct numerical simulation of turbulent channel flow are studied using the methodology described by Aasen and Furuheim (2008). First order critical points inside the field are obtained using a trilinear interpolation scheme. Classification of the critical points found in the three dimensional field is performed by studying individual phase planes of each critical point. Distribution of the critical points and their classifications are compared for different parts of the converging-diverging turbulent channel flow in order to investigate the effect of imposed pressure gradient within the domain. Results obtained from the numerical simulation with Re\textsubscript{θ} = 395 are compared with three-component two-dimensional stereo PIV data recorded over the decelerating part of a converging-diverging bump placed inside the wind tunnel possessing a significantly higher Reynolds number of 19100.

1M. Marquillie et al. (2008), J. Turbulence, vol 9, no 1, pp. 1-23.

2:34PM QB.00009 Bayesian Assessment of Mean Velocity Profile Models in Wall-Bounded Turbulence, ROBERT MOSER, TODD OLIVER, University of Texas — The form of the mean velocity profile in high-Reynolds-number wall-bounded turbulent shear flows has been the subject of renewed interest in recent years. A number of questions have been raised regarding the universality of the von Karman constant, the dependence of the over-lap layer on Reynolds number and even the appropriateness of a logarithmic description of the overlap layer. The questions have been difficult to resolve because the models predict subtle differences in the mean velocity profiles at finite Reynolds number. However, these subtle differences are important for scaling to very high Reynolds number and for inferring wall shear stress when direct measurements are not available. In this work, Bayesian inference is used to infer parameters (e.g. the Karman constant) and their uncertainty in a variety of turbulent mean velocity representations using experimental data over a wide range of Reynolds number. Moreover, an information theory-based multi-model formalism is used to rank competing models (e.g. the standard log and power laws and finite Reynolds number refinements of these profiles) by a metric that naturally balances data fit versus model complexity. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FCS2-08NA28615].
interface oscillations. which matches the Richtmyer constant linear growth-rate. Differences in start-up time and growth rate oscillations are demonstrated to correlate directly to the fluid motions and for both specular and diffuse reflections. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, over the mesh without deforming it in both algorithms. The two algorithms produce essentially identical results (except for noise) for a wide range of piston velocities. The resulting formulation is simple to code and provides considerable computational savings for a wide range of problems of practical interest. Sandia National Laboratories is a multi-program laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

Acknowledgements: The research is supported by Boeing and AFOSR. B.J.M. gratefully acknowledges NSF-CAREER award no. 0747672 (program managers W. W. Schultz & H. H. Winter).

Tuesday, November 23, 2010 12:50PM - 2:47PM
Session QC Rarefied Gases and Compressible Flows
Long Beach Convention Center 102A

12:50PM QC.00001 Importance sampling based direct simulation Monte Carlo method, PRAKASH VEDULA, University of Oklahoma, DUSTIN OTTEN, Lockheed Martin — We propose a novel and efficient approach, termed as importance sampling based direct simulation Monte Carlo (ISDSCMC), for prediction of nonequilibrium flows via solution of the Boltzmann equation. Besides leading to a reduction in computational cost, ISDSCMC also results in a reduction in the statistical scatter compared to conventional direct simulation Monte Carlo (DSMC) and hence appears to be potentially useful for prediction of a variety of flows, especially where the signal to noise ratio is small (e.g. microflows). In this particle in cell approach, the computational particles are initially assigned weights (or importance) based on constraints on generalized moments of velocity. Solution of the Boltzmann equation is achieved by use of (i) a streaming operator which streamlines the computational particles and (ii) a collision operator where the representative collision pairs are selected stochastically based on particle weights via an acceptance-rejection algorithm. Performance of ISDSCMC approach is evaluated using analysis of three canonical microflows, namely (i) thermal Couette flow, (ii) velocity-slip Couette flow and (iii) Poiseuille flow. Our results based on ISDSCMC indicate good agreement with those obtained from conventional DSMC methods. The potential advantages of this (ISDSCMC) approach to granular flows will also be demonstrated using simulations of homogeneous relaxation of a granular gas.

1:03PM QC.00002 Variance-reduced DSMC simulations of low-signal flows1, GREGG RADTKE, HUSAIN AL-MOHSSEN, Mechanical Engineering Department, MIT, MICHAEL A. GALLIS, Sandia National Laboratories, NICOLAS HADJICONSTANTINOU, Mechanical Engineering Department, MIT — We present a variance-reduced direct Monte Carlo method for efficient simulation of low-signal kinetic problems. In contrast to previous variance-reduction methods, the method presented here, referred to as VRDSMC, is able to substantially reduce variance with essentially no modification to the standard DSMC algorithm. This is achieved by introducing an auxiliary equilibrium simulation which, via an importance weight formulation, uses the same particle data as the non-equilibrium (DSMC) calculation. The desired hydrodynamic fields are expressed in terms of the difference between the equilibrium and the non-equilibrium results, which yields drastically reduced statistical uncertainty because it exploits the correlation between the two simulations. The resulting formulation is simple to code and provides considerable computational savings for a wide range of problems of practical interest. Sandia National Laboratories is a multi-program laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

1:16PM QC.00003 ABSTRACT WITHDRAWN –

1:29PM QC.00004 Improved-Efficiency DSMC Collision-Partner Selection Schemes, MICHAEL A. GALLIS, JOHN R. TORCZYNSKI, Sandia National Laboratories — The effect of the collision-partner selection scheme on the accuracy and efficiency of the Direct Simulation Monte Carlo (DSMC) method of Bird is investigated. Three schemes to improve the efficiency of the method are proposed in which the standard random collision-partner selection scheme is replaced with a near-neighbor one. These near-neighbor schemes limit the number of selections used to the nearest-neighbor by a fixed number, a fixed prescribed discretization error, or the distance traveled by the colliding molecule. These three schemes are evaluated for one-dimensional Fourier flow and two-dimensional hypersonic flow over a biconic. Their convergence characteristics as functions of spatial and temporal discretization and the number of simulators per cell are compared to the convergence characteristics of the sophisticated and standard DSMC algorithms. Improved performance is obtained if the population from which possible collision partners are selected is an appropriate subset of the population of the cell. Sandia National Laboratories is a multi-program laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

1:42PM QC.00005 DSMC Algorithms for Moving-Boundary Problems, J.R. TORCZYNSKI, M.A. GALLIS, Sandia National Laboratories — Direct Simulation Monte Carlo (DSMC) algorithms for problems with moving boundaries are investigated. The motivation is a microbeam that moves out-of-plane toward and away from a parallel substrate. For implementation and verification purposes, the simpler but analogous approach is investigated. The computational particles are initially assigned weights (or importance) based on constraints on generalized moments of velocity. The two algorithms produce essentially identical results (except for noise) for a wide range of piston motions and for both specular and diffuse reflections. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

1:55PM QC.00006 A comparison study of planar Richtmyer-Meshkov instability in Mie-Grüneisen fluids and perfect gases1, G.M. WARD, D.I. PULLIN, Caltech — A numerical study of planar Richtmyer-Meshkov instability in fluids with Mie-Grüneisen equations of state is presented and compared to similar perfect gas flows to expose the role of the equation of state. Results for single and triple-mode planar Richtmyer-Meshkov instability, when a reflected shock wave occurs, are first given for MORG and Molybdenum. Comparison is drawn to the standard random collision-partner selection scheme. The desired hydrodynamic fields are expressed in terms of the difference between the equilibrium and the non-equilibrium results, which yields drastically reduced statistical uncertainty because it exploits the correlation between the two simulations. The resulting formulation is simple to code and provides considerable computational savings for a wide range of problems of practical interest. Sandia National Laboratories is a multi-program laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

1Supported by the DoE
Wave superposition, ANDREW HIGGINS, NAVID MEHRJOO, McGill University — Detonation waves in gases are inherently multidimensional due to their cellular structure, and detonations in liquids and heterogeneous solids are often associated with instabilities and stochastic, localized reaction centers (i.e., hot spots). To explore the statistical nature of detonation dynamics in such systems, a simple model that idealizes detonation propagation as an ensemble of interacting blast waves originating from spatially random point sources has been proposed. Prior results using this model exhibited features that have been observed in real detonating systems, such as anomalous scaling between axisymmetric and two-dimensional geometries. However, those efforts used simple linear superposition of the blast waves. The present work uses a model of blast wave superposition developed for multiple-source explosions (the LAMB approximation) that incorporates the nonlinear interaction of shock waves analytically, permitting the effect of a more physical model of blast wave interaction to be explored. The results are suggestive of a universal behavior in systems of spatially randomized energy sources.

Unsteady numerical simulations over the BLOODHOUND supersonic car, GUILLERMO ARAYA, Swansea University, B. EVANS, O. HASSAN, K. MORGAN — One of the main purposes of the BLOODHOUND SSC project consists on achieving the first 1000 mph record on land. In this study, unsteady flow predictions over the BLOODHOUND supersonic car (http://www.bloodhoundssc.swan.ac.uk/), are shown and discussed with Mach numbers up to 1.4. The governing equations of the flow are solved by implementing a hybrid RANS/LES approach. Close to walls the flow is treated with the RANS-equations and the Menter SST model is considered. In zones with separated flows and significant unsteadiness a subgrid-scale stress model is implemented. The results are suggestive of a universal behavior in systems of spatially randomized energy sources.

Shock-initiated Combustion of a Spherical Density Inhomogeneity, NICHOLAS HAEHN, JASON OAKLEY, DAVID ROTHAMER, MARK ANDERSON, UW - Madison, DEVESH RANJAN, Texas A&M University, RICCARDO BONAZZA, UW - Madison — A spherical density inhomogeneity is prepared using fuel and oxidizer at a stoichiometric ratio and Xe as a diluent that increases the overall density of the bubble mixture (55% Xe, 30% H₂, 15% O₂). The experiments are performed in the Wisconsin Shock Tube Laboratory in a 9.2 m vertical shock tube with a 25.4 cm × 25.4 cm square cross-section. An injector is used to generate a 5 cm diameter soap film bubble filled with the combustible mixture. The injector retracts flush into the side of the tube releasing the bubble into a state of free fall. The combustible bubble is accelerated by a planar shock wave in N₂ (2.0 < M < 2.8). The mismatch of acoustic impedances results in shock-focusing at the downstream pole of the bubble. The shock focusing results in localized temperatures and pressures significantly larger than nominal conditions behind a planar shock wave, resulting in auto-ignition at the focus. Planar Mie scattering and chemiluminescence are used simultaneously to visualize the bubble morphology and combustion characteristics. During the combustion phase, both the span-wise and stream-wise lengths of the bubble are seen to increase compared to the non-combustible scenario. Additionally, smaller instabilities are observed on the upstream surface, which are absent in the non-combustible bubbles.

Turbulence Simulations V — Session QD.12550PM - 2:21PM — Long Beach Convention Center 102B

Conditional flow structures in a Large Eddy Simulation of the fully developed wind-turbine array boundary layer1, CLAIRE VERHULST, CHARLES MENEVEAU, Johns Hopkins University, Baltimore, MD — Wind-turbines deployed in a large array experience a decrease in individual efficiency due to interactions among themselves and the atmospheric boundary layer (ABL). A fully developed flow regime can be established when this wind-turbine array is an order of magnitude longer than the height of the ABL. Under this condition, vertical entrainment of kinetic energy is essential for power extraction. In order to characterize this entrainment process, a Large Eddy Simulation of the fully developed wind-turbine atmospheric boundary layer (WTABL) is performed using a pseudo-spectral method with periodic boundary conditions in the horizontal directions. The wind-turbines are modeled as drag disks with a force proportional to the local disk-averaged velocity (Calaf et al. 2010, Phys. Fluids 22, 015110). Conditional averaging of the WTABL velocity field based on thresholds set on the instantaneous power extraction is performed to determine conditional coherent flow structures associated with large values of power extraction. Properties of the conditional structures are examined and their dependencies on WT loading factors are studied.

Conditional flow structures in a Large Eddy Simulation of the fully developed wind-turbine array boundary layer1, STIMIT SHAH, ELIE BOU-ZEID, Princeton University — Large Eddy Simulations (LES) of the atmospheric boundary layer (ABL) are performed using a recently developed dynamic subgrid- scale (SGS) model. The model calculates both the Smagorinsky coefficient and the SGS Prandtl number dynamically based on the Lagrangian scale-dependent model in which required averages are accumulated in time, following fluid trajectories of the resolved velocity field. Simulations for both stable and unstable atmospheric boundary layers with heterogeneous surface fluxes are carried out to investigate the effect of surface variability on the turbulent kinetic energy budget and mixing in the ABL, with a special emphasis on the implications for turbulent transport similarity and turbulence closure in coarse atmospheric models.

Large Eddy Simulation of stable and unstable atmospheric boundary layers over heterogeneous terrain1, STIMIT SHAH, ELIE BOU-ZEID, Princeton University — Large Eddy Simulations (LES) of the atmospheric boundary layer (ABL) are performed using a recently developed dynamic subgrid- scale (SGS) model. The model calculates both the Smagorinsky coefficient and the SGS Prandtl number dynamically based on the Lagrangian scale-dependent model in which required averages are accumulated in time, following fluid trajectories of the resolved velocity field. Simulations for both stable and unstable atmospheric boundary layers with heterogeneous surface fluxes are carried out to investigate the effect of surface variability on the turbulent kinetic energy budget and mixing in the ABL, with a special emphasis on the implications for turbulent transport similarity and turbulence closure in coarse atmospheric models.

Supported by the Energy Grand Challenges Program of Princeton University.
Applying the v2f and the algebraic structure-based Reynolds stress closures to wind flow over complex terrain. The quark and Gluons are made up of the system of quark confinement. Not only thing has the mass-energy of itself, the frequence of wave, and cannot represent turbulent anisotropy. In other applications the v2f turbulence closure has shown a good ability to predict flow separation. Similarly the algebraic structure-based model (ASBM) has shown promise in capturing turbulent anisotropy. The flow over a representative hill which includes these features is calculated using the RANS equations with both the v2f and ASBM closures. A novel implementation of the ASBM closure is developed allowing a stable solution to be obtained. The results are compared with experimental data for the same flow and a good agreement is obtained for the separated region and the Reynolds stress components. Wall functions are developed for the v2f closure to enable the simulation of higher Reynolds number flows that include surface roughness. The results are compared with experimental data and accurately capture the separated region.

Optimization of turbine spacing in the fully developed wind turbine array boundary layer. We consider the fully developed wind turbine array boundary layer, which is a regime of relevance when wind farms exceed the height of the atmospheric boundary layer by over an order of magnitude. Based on extensive LES studies of such boundary layers, a simple physics-based parameterization of the effective surface roughness was developed [see Calaf et al. Phys. Fluids 22, 015110 (2010)]. The model depends upon wind turbine spacing, height, loading factors, ground roughness, etc. Using this model for induced surface roughness of large wind-farms, we proceed to establish optimal spacings between wind turbines, considering constant imposed geostrophic wind forcing. We examine the dependence of the optimal spacing on the ratio between cost of wind turbine and land surface. We find that optimal average turbine spacing typically is between ten and twenty rotor diameters. This spacing is considerably higher than that used in conventional wind farms.

The LES of the channel flow in a non aligned system of coordinates. The plane channel flow continues to be a very important test case for the verification and the validation of LES. In the channel flow test there is a privileged direction, usually one reference axis is oriented along the stream and the size of the computational box is increased in the streamwise direction, which is a regime of relevance when wind farms exceed the height of the atmospheric boundary layer by over an order of magnitude. Based on extensive LES studies of such boundary layers, a simple physics-based parameterization of the effective surface roughness was developed [see Calaf et al. Phys. Fluids 22, 015110 (2010)]. The model depends upon wind turbine spacing, height, loading factors, ground roughness, etc. Using this model for induced surface roughness of large wind-farms, we proceed to establish optimal spacings between wind turbines, considering constant imposed geostrophic wind forcing. We examine the dependence of the optimal spacing on the ratio between cost of wind turbine and land surface. We find that optimal average turbine spacing typically is between ten and twenty rotor diameters. This spacing is considerably higher than that used in conventional wind farms.

Isogeometric Variational Multiscale Large Eddy Simulation of Turbulent Flow through Annulus Channel. The null geodesic congruence for the Lorentzian version of Hawking’s wormhole is studied, in spherical Rindler coordinates. One finds that the wormhole throat, where the stress energy is mostly located, expands exponentially and the flare-out condition is satisfied. The expanding fluid is anisotropic, the mean pressure gradient, and the homogeneities of the Reynolds stresses are destroyed. In our paper we simulate the channel flow in a rotated system of coordinates, and we compare the results with the stream aligned data. We think that this test could evidence the flexibility of different LES codes and LES subgrid models to simulate the turbulent flow and to capture the correct statistical values in non aligned conditions. The first preliminary results are slightly contradictory: the resolved Reynolds stresses seem degraded while the mean flow is better predicted. The dynamic anisotropic subgrid model of Abbà, Cercignani and Vallettaro seems well fitted to represent correctly the large scales in non aligned conditions.

Light dragging phenomenon and expanding wormholes. The null geodesic congruence for the Lorentzian version of Hawking’s wormhole is studied, in spherical Rindler coordinates. One finds that the wormhole throat, where the stress energy is mostly located, expands exponentially and the flare-out condition is satisfied. The expanding fluid is anisotropic, and has a mean pressure that is one third of the energy density, as for a null fluid. A time reversal is equivalent with an inversion applied to the radial coordinate. Far from the throat (the light cone in Cartesian coordinates) the energy density of the fluid no longer depends on the Newton constant G and acquires an exponential grow. The wormhole throat, where the stress energy is mostly located, expands exponentially and the flare-out condition is satisfied. The expanding fluid is anisotropic, and has a mean pressure that is one third of the energy density, as for a null fluid. A time reversal is equivalent with an inversion applied to the radial coordinate. Far from the throat (the light cone in Cartesian coordinates) the energy density of the fluid no longer depends on the Newton constant G and acquires an exponential grow.
with a mean diameter less than 10 microns in diameter. This work is supported by NIH grant R01EB006476.

Current methods to produce PFC microdroplets, such as high speed shaking or sonication, result in polydisperse droplet distributions where a fraction of liquid perfluorocarbon (PFC) microdroplets are vaporized using focused ultrasound to form gas bubbles that are approximately 125 times larger in volume. Gas,...

1:16PM QE.00003 Seeing Fluid Physics: Outcomes From a Course on Flow Visualization, JEAN HERTZBERG, University of Colorado, Boulder — Since 2003, a course on flow visualization has been offered to mixed teams of engineering and fine arts photography students at the University of Colorado. The most significant outcome of the course is the impact on students’ perceptions; they see fluid physics as ubiquitous in the environment after taking the course. A survey instrument has been developed that explores student perceptions of fluid physics, and has been administered to students in the flow visualization course, and in a traditional junior level fluid mechanics course. Survey results indicate that the students in the flow visualization course notice fluid physics in daily life at an increased rate, and their attitude (affect) towards fluids is improved compared to students in the traditional course. The use of photography in improving student perceptions is being extended to a course on perception of design; preliminary results from a survey on attitudes towards design will be presented. Examples of student images from both courses will be presented as well.

1:29PM QE.00004 New Ways of Teaching Upper-division courses: Descriptions and Results, RACHEL PEPPER, STEPHANIE CHASTEEN, STEVEN POLLOCK, MICHAEL DUBSON, PAUL BEALE, KATHERINE PERKINS, University of Colorado — Over the past three years, the physics faculty at the University of Colorado have worked to transform two core courses in our upper-division undergraduate physics curriculum, Electricity and Magnetism and Quantum Mechanics, with the goals of (a) improving student learning and (b) developing materials and approaches that other faculty may adopt or adapt to their teaching environment. The transformation of our upper-division physics courses may serve as a model for transformation of other upper-division courses, such as fluid mechanics courses. This work began with faculty working groups meeting regularly to define explicit course learning goals. These learning goals served as the foundation for the course transformations that applied the principles of active engagement and learning theory to these upper-division courses. The development of the full curriculum was guided by the results of observations, interviews, and analysis of student work. In this talk, we will outline the reforms – including consensus learning goals, “clicker” questions, tutorials, modified homeworks, and more – and present evidence of the effectiveness of these reforms relative to traditional courses. Some research-based fluid mechanics instructional materials will also be discussed. All of our curriculum materials are available at http://www.colorado.edu/sei/departments/physics.htm.

1:42PM QE.00005 Oscillatory motion of flat square wall-hinged winglets inside a turbulent boundary layer, AMIR ELZAWAWY, Graduate Center of CUNY, YIANNIS ANDREOPOULOS, City College of CUNY — An experiment in a wind tunnel has been designed to investigate the augmented force generation acting on winglets during periodic rotation between zero and ninety degrees angle to the flow. Square and triangular flaps hinged at the wall beneath the flow have been used which were rotated with angular velocities between 10 and 150 rad/s. Strouhal numbers between 0.05 and 1.1 and Stokes numbers between 6300 and 95000 were achieved. Time-resolved Particle Image Velocimetry was implemented by using a continuous laser and fast frame-rate camera to provide qualitative and quantitative information of the flow field. The dynamic lift and drag force coefficients during the periodic motion of the winglet are different than the corresponding coefficients under stationary conditions at the same deployment angle after adjusting for inertial effects. These effects are enhanced with increasing Strouhal number and decrease with increasing boundary layer thickness. A highly intermittent thin boundary layer developing over the forward moving surface of the winglet separates into a shear layer which wraps around to form a large scale vortex which is causing the force augmentation.

1:55PM QE.00006 Combined experimental and computational investigation of sterile airflows in surgical environments, JAMES MCNEILL, JEAN HERTZBERG, ZHIQIANG ZHAI, University of Colorado — Surgical environments in hospitals utilize downward, low-turbulence, sterile air flow across the patient to inhibit transmission of infectious diseases to the surgical site. Full-scale laboratory experiments using particle image velocimetry were conducted to investigate the air distribution above the patient area. Computational fluid dynamics models were developed to further investigate the air distribution within the operating room in order to determine the impact of ventilation design of airborne infectious disease pathways. Both Reynolds-averaged Navier-Stokes equations and large eddy simulation techniques are currently being used in the computational modeling to study the effect of turbulence modeling on the indoor air distribution. CFD models are being calibrated based on the experimental data and will be used to study the probability of infectious particles entering the sterile region of the room.

This research is supported by ASHRAE RP1397.

2:08PM QE.00007 The incorporation of computational fluid dynamics (CFD) capabilities with RISK, an indoor air quality zonal model developed by the U.S. EPA, DAVID MARR, U.S. EPA — Individual exposure to indoor contaminant concentrations is often estimated using assumptions of fully mixed conditions. The applicability of such an assumption can vary significantly based on the ventilation design and contaminant of interest. To solve for gradients in the contaminant concentration field, a CFD solver has been added to RISK, the primary indoor air quality model developed by the U.S. Environmental Protection Agency. The RISK model was created to solve for “zonal” concentrations based on emission characteristics of indoor materials and emission sources. Current updates to this model allow for a greater resolution and therefore more detailed view of risk and exposure in the indoor environment towards risk management. CFD results are compared to particle image velocimetry (PIV) experimental databases acquired at the U.S. EPA and Syracuse University. This presentation includes a brief overview of the model capabilities, steps towards validation of the model output, and examples of indoor contaminant transport from common indoor material emissions.

2:21PM QE.00008 Flow-Induced Stress Distribution in Porous Scaffolds, DIMITRIOS PAPAVASSILIOU, ROMAN VORONOV, SAMUEL VANGORDON, VASSILIOS SIKAVITSAS, The University of Oklahoma — Flow-induced stresses help the differentiation and proliferation of mesenchymal cells cultured in porous scaffolds within perfusion bioreactors. The distribution of stresses in a scaffold is thus important for understanding the tissue growth process in such reactors. Computational results for flow through Poly-L-Lactic Acid porous scaffolds that have been produced with salt-leaching techniques, and for scaffolds that have been constructed with nonwoven fibers, indicate that the probability density function (pdf) of the wall stress, when normalized with the mean and the standard deviation of the pdf, appears to follow a single type of pdf. The scaffolds were imaged with micro-CT and the simulations were run with lattice Boltzmann methods. The parameters of the distribution can be obtained using Darcy’s law and the Blake-Kozeny-Carman equation. Experimental results available in the literature appear to corroborate the computational findings, leading to the conclusion that stresses in high-porosity porous materials follow a single distribution.

2:34PM QE.00009 Microfluidic Production of Monodisperse Perfluorocarbon Microdroplets, DAVID LI, KEVIN SCHALTE, J. BRIAN FOWLKES, JOSEPH BULL, University of Michigan — Acoustic droplet vaporization (ADV) is process in which liquid perfluorocarbon (PFC) microdroplets are vaporized using focused ultrasound to form gas bubbles that are approximately 125 times larger in volume. Gas embolotherapy is a novel cancer treatment that uses ADV in vivo to strategically form gas emboli, which can lodge in the microcirculation and starve tumors. Current methods to produce PFC microdroplets, such as high speed shaking or sonication, result in polydisperse droplet distributions where a fraction of droplets fall within the 2-10 microns range. In the clinical application with such a droplet distribution, large droplets are filtered by the lungs and small droplets result in bubbles that are too small to lodge in the tumor vasculature. Consequently, there is a need for a monodisperse droplet distribution. A microfluidic based device has been developed in order to produce such monodisperse PFC microdroplets. The device used hydrodynamic flow focusing to create droplets with a mean diameter less than 10 microns in diameter. This work is supported by NIH grant R01EB006476.
The performance of diffusers designed for optimum pressure recovery is governed by flow separation which can be very sensitive to inlet perturbations. We are examining the effect of upstream disturbances on the performance of practical annular diffusers. Experiments are conducted in an annular diffuser sector containing a single NACA 0015 airfoil shaped support strut. Three component, time averaged velocities are measured using magnetic resonance velocimetry and static pressure data are measured with conventional wall taps. We are testing four inlet conditions: a uniform velocity profile with thin boundary layers and relatively low turbulence intensity, a similar case with higher turbulence levels, a mean profile with uniform velocity except for a high velocity wall jet at the outer radius, and a nonuniform profile in which the mean velocity decreases with increasing radius. Generally, the results show that the diffuser acts to increase flow distortion. For the case with the radial velocity gradient, passing through the diffuser strongly increases the velocity gradient. The wall jet on the outer (diluting) wall eliminates flow separation resulting in higher pressure recovery and thicker wall boundary layers on the other three walls. Interestingly, the separated wake of the support strut closes more rapidly for the case with the radial velocity gradient.

**Tuesday, November 23, 2010 12:50PM - 3:00PM –**

**Session QF Non-Newtonian Flows II** Long Beach Convention Center 103A

**1:03PM QF.00002 Symmetric factorization of the conformation tensor in viscoelastic fluid models**, BECCA THOMASES, University of California, Davis, Dept. of Mathematics, NUSRAT BALCI, Institute for Mathematics and its Applications, University of Minnesota, Minneapolis, MICHAEL RENARDY, Virginia Tech, Dept. of Mathematics, CHARLES DOERING, University of Michigan, Ann Arbor, Dept. of Physics and Mathematical — The positive definite symmetric polymer conformation tensor possesses a unique symmetric square root that satisfies a closed evolution equation in the Oldroyd-B and FENE-P models of viscoelastic fluid flow. When expressed in terms of the velocity field and the symmetric square root of the conformation tensor, these models’ equations of motion formally constitute an evolution in a Hilbert space with a total energy functional that defines a norm. Moreover, this formulation is easily implemented in direct numerical simulations resulting in significant practical advantages in terms of both accuracy and stability.

**1:16PM QF.00003 Slow motion and deformation of a viscoplastic drop in a viscous fluid**1, OLGA LAVRENTEVA, IRINA SMAGIN, AVINOAM NIR, Chemical Engineering Dept., Technion — The slow sedimentation of a deformable viscoplastic drop in a Newtonian fluid is studied making use of a variation of integral equation method. The Green function for the Stokes equation is used and the non-Newtonian stress is treated as a source term. The computations carried out for a range of physical parameters of the system revealed that increasing in the yield stress magnitude (the Bingham number, Bn) stabilizes both oblate and prolate drops. This is in contrast to the effect of the viscosity of Newtonian drop that is known to destabilize oblate drops. Strong stabilization effect can be explained by the presence of unyielded zones inside falling drops. An interesting observation is that the growth of the limiting viscosity of the Bingham fluid destabilize oblate deformations at low Bn and have stabilizing effect at higher Bn.

1This research was supported by The Israel Science Foundation (grant 847/06).

**1:29PM QF.00004 Steady State Visco-Elastic Rimming Flow**, ANTON MAZURENKO, Massachusetts Institute of Technology, SERGEI FOMIN, CSU Chico, BRENTH NELSON, University of Illinois, Urbana-Champaign, JARED DEBRUNNER, CSU Chico — Using scale analysis and the method of perturbations, a theoretical description is obtained for the steady-state non-Newtonian fluid on the inner wall of the rotating horizontal cylinder. The Maxwell upper-convective equation is chosen to model the visco-elastic properties of the fluid. In the general case, the derived governing equations can be solved only numerically. However, since the polymeric solutes used in roto-molding and coating technologies exhibit the relatively weak elastic properties, the Deborah number for such flows is rather small (De<1). Exploiting this fact, the perturbation method is applied for simplification of the model. As a result, the first order non-linear differential equation for the thickness of the fluid film is derived. An approximate analytical solution of this equation is found. The accuracy of analytical solution is verified by the direct numerical solution of the derived equation. The obtained equation is rather complex and contains several critical points. These points are classified by the analysis of the corresponding autonomous system. The type and location of these critical points are accounted for during numerical solution of the equation. Using the obtained solutions, the criteria which guarantee the stable steady-state flow of the liquid polymer and the uniform final thickness of the coating film are determined. The bounds for the different flow regimes and principal controlling parameters are identified.

**1:42PM QF.00005 Effect of Polymer Additives on Heat Transfer in a Laminar Boundary-Layer Flow**, EMILY S.C. CHING, The Chinese University of Hong Kong, ROBERTO BENZI, Universita Tor Vergata, VIVIEN W.S. CHU, The Chinese University of Hong Kong — We have carried out a theoretical analysis of the effect of polymer additives on heat transfer in a laminar boundary-layer flow. We consider the simple Oldroyd-B model of polymers and show that the effect of the polymers can be understood as a position-dependent effective viscosity. We find that the presence of polymers leads to a reduction in the Nusselt number (Nu), the dimensionless number measuring heat transport. Moreover, the extent of reduction increases with the concentration of the polymers. We shall also discuss the relevance of our work to the recent experimental observation of a decrease in Nu in turbulent thermal convection upon the addition of polymers.

1We acknowledge support by the Hong Kong Research Grants Council (grant no. CUHK 400708).

**2:04PM QF.00006 Purely elastic instabilities in parallel shear flows**, LICHAO PAN, PAULO ARRATIA, University of Pennsylvania — In this talk, the stability of viscoelastic fluids in parallel shear flow at low Reynolds number (Re<0.01) is experimentally investigated using dye advection visualization and particle tracking velocimetry. The fluid of interest is a dilute polymeric fluid with nearly constant shear-viscosity. The experimental setup is a microchannel that is 3 cm long and 100 µm wide. The channel consists of two regions. The first region contains a linear array of cylinders designed to introduce perturbations to the viscoelastic flow. The second region is a long (2.7 cm) and straight channel devoid of cylinders in which the spatial-and temporal behavior of the initial perturbation is monitored. This second region is the parallel shear geometry. Preliminary results based on velocity measurements show that the initial disturbance is sustained far downstream in the parallel shear geometry above certain Wissenberg number (Wi), and increase non-linearly with Wi even at vanishing small Re. For the viscoelastic fluid, curved streamlines are observed in the parallel shear geometry region of the channel. No velocity fluctuations or curved streamlines are found for the Newtonian fluid under the same conditions.
2:08PM QF.00007 Rinsing Flows of Non-Newtonian Fluids , GERALD FULLER, TRAVIS WALKER, TIENYI HSU, Stanford University, PATRICK ANDERSON, Eindhoven University of Technology — The rinsing flow of a jet of water impinging onto both Newtonian and non-Newtonian viscous fluids has been considered to qualitatively and quantitatively understand the flow structure of the resulting hydraulic jump. This study seeks to investigate the interactions of the two fluid system during the transient growth of the flow profile. This growth is seen to vary drastically in magnitude, velocity, and topology, while undergoing varying instabilities, depending on the properties of the coating fluid. Currently, four classes of test fluids, all having approximately equal viscosities at low shear, have been chosen for this study: a Newtonian solution, a viscoelastic polymer solution, a Boger fluid, and a worm-like micelle solution. Each fluid experiences Saffman-Taylor instabilities, and the experiments show that the elasticity of the samples will influence the pattern of the instabilities. The elasticity is also seen to dampen the disturbances of the hydraulic jump, influence the overall jump height, and vary the radial growth of the jump. In addition, the shear-thinning nature of the samples seems to influence the overall velocity of the radial growth, while determining the geometry of the driving front. Finite element simulations are also presented in an attempt to understand these complex flow kinematics.

2:21PM QF.00008 Impact of a cylindrical rod on a concentrated particle suspension: dynamics, crack growth and relaxation , EGLIND MYFTIU, MATTHEIU ROCHE, PILNAM KIM, HOWARD A. STONE, Mechanical and Aerospace Engineering, Princeton University — Many highly concentrated particle suspensions are shear thickening; the viscosity increases with shear rate. The physics underlying shear thickening is still under discussion. In recent years, it was pointed out that shear thickening may be connected with a liquid-to-solid phase transition of the suspension. We provide direct evidence of this transition by studying the behavior of aqueous cornstarch suspensions of various concentration and layer thicknesses after impact of a free-falling cylindrical rod, which induces high strain rates and stresses. We observe patterns of regularly distributed radial cracks growing outwards from the impact region. Just after impact, a wave propagates on the surface of the layer and in the neighborhood of the impact a cavity expands. During this expansion, the cavity boundary is torn, and cracks start to grow. These cracks have rough boundaries, as is seen in solids. Once the cracks have reached their maximal extension, the suspension relaxes. The solvent slowly fills the cracks, until the layer returns to its initial shape. We discuss the influence of the layer thickness, starch concentration and impact energy on the dynamics of these cracks. We also discuss some properties of the solid phase of these suspensions as well as their relaxation dynamics.

2:34PM QF.00009 Pinch-off Dynamics of Non-Newtonian Fluids , F.M. HUISMAN, S.R. GUTMAN, P. TABOREK, University of California-Irvine — The pinch-off dynamics of a variety of shear-thinning fluids (foams, concentrated emulsions, and slurries) were studied using high speed videography. The pinch was characterized by the variation of the minimum neck radius rmin as a function of the time to pinch t, with rmin prop η^n. The rheology of shear thinning fluids can be characterized by an exponent n = kη^κ, with n < 1. We found that for a variety of shear-thinning fluids including mayonnaise and acrylic paint, rmin scales with t to a power α equal to the flow index for the particular fluid. The flow index was measured using a TA instruments AR-G2 rheometer. The flow index for acrylic paint was 0.440 +/- 0.014 and rmin scales with t to the 0.41 +/- 0.03; for mayonnaise the flow index was 0.355 +/- 0.014; and rmin scales with α to the 0.35 +/- 0.02. To study the transition from conventional Newtonian pinch, we systematically varied the concentration of a water-Xanthan gum mixture.

2:47PM QF.00010 Buckling of cornstarch solutions after pinch-off: evidence for a jamming transition at high extensional rates , MATTHIEU ROCHE, OYKU M. AKKAYA, Mechanical and Aerospace Engineering, Princeton University, HAMID KELLAY, Centre de Physique Moléculaire Optique et Hertzienne, Universite Bordeaux 1, France, HOWARD A. STONE, Mechanical and Aerospace Engineering, Princeton University — We studied the behavior of density-matched cornstarch solutions during and after pinch-off from a needle. We observed an exponential slowing down in the thinning dynamics of the bridge connecting the droplet to the needle during which the bridge adopts a cylindrical shape. At this stage, the flow is mainly extensional allowing us to explore the behavior of starch solutions at extension rates greater than 10 s^-1. The bridge continues to thin and then destabilizes leading to break-up in multiple parts. These parts retract on themselves and buckle. We show that this buckling behavior can be understood as a consequence of a liquid-to-solid transition of starch solutions during thinning. Using microscopy, we demonstrate that the neck is inhomogeneous during the last stages of pinch-off: the thinner sections of the neck are fluid while the thicker regions are jammed. We explain buckling by showing that the bridge deforms around its fluid sections, making this system analogous to a chain of solid links connected by fluid bridges.

Tuesday, November 23, 2010 12:50PM - 3:00PM –
Session QG Stratified Flows I Long Beach Convention Center 103B

12:50PM QG.00001 Interleaving intrusions between adjacent layered stratifications1, BENJAMIN MAURER, Scripps Institution of Oceanography, PAUL LINDEN, DAMTP, University of Cambridge — Interfacial gravity currents occur when horizontal density gradients result in the intrusion of a fluid along the interface between two layers of ambient fluid. This system has traditionally been studied in the case where the ambient fluid consists of only two layers, however, many oceanographic and atmospheric flows involve the interleaving adjacent stratifications consisting of multiple layers. We present an experimental and numerical study of the interleaving of multiple interfacial intrusions propagating in both directions. For the simple case of two adjacent layered stratifications where the average density of both sides is equivalent, the adjacent layers interleave at uniform speeds. However, if the average density of one side is increased, the individual current speeds show a marked departure from the speeds predicted from local initial geometry. Finite element simulations are also presented in an attempt to understand these complex flow kinematics.

1This research was supported by National Science Foundation grant CTS 0756396

1:03PM QG.00002 Intrusive gravity currents and the solitary wave lifecycle in a cylindrical geometry , JUSTINE McMILLAN, BRUCE SUTHERLAND, University of Alberta — An “intrusive gravity current” or “intrusion” arises when a fluid of one density propagates at an intermediate depth within a stratified ambient. Numerous experimental and theoretical studies have examined the propagation of these currents in a rectilinear geometry, however, the dynamics of radially spreading axisymmetric intrusive gravity currents is less well established. By way of full-depth lock release experiments and numerical simulations, we examine the propagation of vertically symmetric intrusive gravity currents in a two-layer ambient in a cylindrical geometry. We show that the strong stratification at the interface supports the formation of a mode-2 solitary wave that surrounds the intrusion head and carries it outwards at a constant speed beyond 6 lock radii. The wave and intrusion propagate faster than a linear long wave; therefore, there is strong evidence to support that the wave is indeed nonlinear. By extending rectilinear KdV theory to allow the wave amplitude to decay as r^−p with p ≈ ½, we show that from a single measurement of wave amplitude, the theory can be used to accurately predict the amplitude, speed and spread of the wave during its nonlinear evolution phase.
Horizontal Convection
KATARYNA MATUSIK, STEFAN LLEWELLYN SMITH, UCSD MAE Dept. — Horizontal convection, caused by differential heating along the horizontal boundary of a fluid, is a model of the meridional overturning circulation of the oceans. We explore aspects of horizontal convection using laboratory experiments. We use salt rather than heat, with sinks and sources of dense and fresh water on the upper boundary of the tank so that the net flux of salt into the tank is statistically zero. We measure the density using the Synthetic Schlieren method and a conductivity probe, and dye the incoming fluid and measure its concentration using optical methods. We also use particle tracking to visualize the velocity field within the tank. Our goal is to examine the role of the aspect ratio and governing dimensionless parameters of the system, as well as the effect of the location of sources and sinks, and relate these to features of the flow such as boundary-layer thickness and net overturning circulation. The use of salt rather than temperature results in a high Schmidt number, with implications for the understanding of the experiments and their relation to the ocean.

L/D > m → ∞ and the behaviour is characterised solely by Fr_L/D. We find that the releases are sensitively dependent upon Fr_L/D and L/D and three rise height regimes, ‘the weak fountain regime’, ‘the vorticity development regime’ and ‘the forced release regime’, are identified.

1:29PM QQ.00004 Lattice Boltzmann simulation of buoyancy-driven flow of two immiscible fluids in an inclined channel. S.P. VANKA, University of Illinois at Urbana-Champaign, K.C. SAHU, Indian Institute of Technology, Hyderabad — Buoyancy-driven flow due to unstable density stratification of two immiscible fluids in a confined inclined channel is studied by Lattice Boltzmann method using the color segregation approach followed by Wu et al. (Int. J. Numer. Mech. Fluids, 2008, 57, 793-810) and Lishchuk et al. (Phys. Rev.E, 2008, 77, 036702). Initially, the upper and lower halves of the channel are filled with heavier and lighter fluids, respectively. In this system Rayleigh-Taylor instability would occur for any perturbation along the “interface” of the two fluids. The fingers of the heavier and lighter fluids then propagate in the downward and upward directions, respectively. The results are compared with earlier experimental and theoretical studies. The spatio-temporal evolution of flow structures and propagation of the finger tips for different angles of inclination and different channel aspect ratio are investigated.

1:42PM QQ.00005 ABSTRACT WITHDRAWN —

1:55PM QQ.00006 Turbulence-induced secondary motion in a buoyancy-driven flow in a circular pipe. JACQUES MAGANUDET, YANNICK HALLEZ, IMFT/Cnrs — We analyze the results of a direct numerical simulation of the turbulent circular pipe between the transverse components of the turbulent velocity fluctuations. Although of small magnitude compared to the main flow, this secondary mean motion is found to exhibit nonzero secondary velocity components in the tube cross section, resulting in a four-cell pattern in the tube cross-section. We analyze the generating mechanism of this secondary flow which turns out to be due to the combined effect of the lateral wall and the shear-induced anisotropy of the mean flow. In the central region located midway between the endwalls, where the turbulent flow is fully developed, the resulting mean flow is found to exhibit nonzero secondary velocity components in the tube cross section, resulting in a four-cell pattern in the tube cross-section. We analyze the generating mechanism of this secondary flow which turns out to be due to the combined effect of the lateral wall and the shear-induced anisotropy of the mean flow. In the central region located midway between the endwalls, where the turbulent flow is fully developed, the resulting mean flow is found to exhibit nonzero secondary velocity components in the tube cross section, resulting in a four-cell pattern in the tube cross-section. We analyze the generating mechanism of this secondary flow which turns out to be due to the combined effect of the lateral wall and the shear-induced anisotropy of the mean flow. In the central region located midway between the endwalls, where the turbulent flow is fully developed, the resulting mean flow is found to exhibit nonzero secondary velocity components in the tube cross section, resulting in a four-cell pattern in the tube cross-section.
An ice sheet that spreads into an ocean is forced to bend due to its buoyancy, and delaminate from the ground to form an ice shelf. The location of the transition between the grounded sheet and the ungrounded shelf is defined as the grounding-line. The dynamics of grounding lines may have a critical effect on the stability of ice sheets, and determining the position of those free boundaries requires additional conditions that can be hard to constrain. We model such a sheet-shelf system as an elastic sheet, partially resting on an elastic substrate and partially resting on an ocean. Solving the two parts simultaneously by matching moments across the grounding line, we can avoid imposing an explicit boundary condition at the grounding line. In the limit of very stiff ground, a Stefan-like boundary condition can be identified at the grounding line, which simplifies the problem into solving only for the shelf. We present a good agreement between our theoretical predictions and laboratory experiments made using thick elastic sheets and a dense salt solution. We evaluate the variation of grounding line position with bed stiffness numerically, and estimate the relevance of an elastic response to viscous ice sheets.

We discuss the influence of settling particles on heat transport within suspensions. We focus on particles that equilibrate by the simulations are in good agreement with previous models and available measurements. Specifically, this work considers the effect on the fluxes of the stability parameter $Ra/R_0$, where $Ra$ is the Rayleigh number and $R_0$ is a characteristic length scale. The ratio $Ra/R_0$ can be seen as a ratio of destabilizing and stabilizing effects. Trends predicted by the simulations are in good agreement with previous models and available measurements.

We present a numerical study of incremental spin-up of a thermally-stratified fluid enclosed within right circular cylinder/annulus with rigid bottom and side walls and stress-free upper surface. This investigation reveals a feasibility for transition from an axisymmetric initial circulation to non-axisymmetric flow patterns, and it is motivated by the desire to compare the spin-up for Dirichlet and Neumann thermal boundary conditions. The numerical simulations demonstrate that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

An experimental site is being developed at the under-construction New Doha International Airport (NDIA) in Qatar to obtain simultaneous measurements of atmospheric wind and ocean surface wave parameters. The site is located at the end of an aircraft approach light line which is about 500 m into the sea. Water at this location is about 6-m deep and ocean floor is almost flat terrain in all directions. The necessary measurement will be accomplished by synchronized operation of three sonic anemometers and two CCD cameras. An overview of the experimental site, along with preliminary data showing the weather trend and the feasibility of extending the study is reported.

An ice sheet that spreads into an ocean is forced to bend due to its buoyancy, and delaminate from the ground to form an ice shelf. The location of the transition between the grounded sheet and the ungrounded shelf is defined as the grounding-line. The dynamics of grounding lines may have a critical effect on the stability of ice sheets, and determining the position of those free boundaries requires additional conditions that can be hard to constrain. We model such a sheet-shelf system as an elastic sheet, partially resting on an elastic substrate and partially resting on an ocean. Solving the two parts simultaneously by matching moments across the grounding line, we can avoid imposing an explicit boundary condition at the grounding line. In the limit of very stiff ground, a Stefan-like boundary condition can be identified at the grounding line, which simplifies the problem into solving only for the shelf. We present a good agreement between our theoretical predictions and laboratory experiments made using thick elastic sheets and a dense salt solution. We evaluate the variation of grounding line position with bed stiffness numerically, and estimate the relevance of an elastic response to viscous ice sheets.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.

A study of atmospheric wind and ocean surface wave interaction is presented. The exchange of momentum between the wind and the ocean surface is the primary source of various oceanic phenomena, both in large and small-scales. Upon reviewing the existing literature, there seems to be a missing link between the standard atmospheric and oceanic-circulation models. We obtain a set of linear integro-differential equations for a proper model, given that the destabilizing mechanism is not purely baroclinic, but that vertical and horizontal shears may contribute to the instability. By characterizing the azimuthal instabilities without introducing any simplification we were able to assess to what extent an insulating boundary condition changes the time-dependent emergence of the instability. Our results agree with previous experimental data and provide a framework for understanding the role played by the baroclinic vorticity in the development of instabilities in thermally-stratified incremental spin-up flows.
2:08PM QH.00007 PSI of the oceanic internal tide, JEROEN HAZEWINKEL, YUE-KIN TSANG, KRAIG B. WINTERS, WILLIAM R. YOUNG, University of California, San Diego, INTERNAL WAVES ACROSS THE PACIFIC COLLABORATION — The dynamics of a forced, low-mode oceanic internal tide propagating poleward on a beta plane are investigated numerically. We focus on the instability that transfers energy from the forced wave to waves at subharmonic frequency near the critical latitude where the subharmonic and local inertial frequencies match. Through parametric subharmonic instability (PSI), energy is transferred to motions that have a fine-scale modal structure, both in the horizontal and vertical directions. During the exponential growth phase, these motions do not provide a major sink of energy for the tide. A significant reduction in poleward energy flux is only observed once the amplitude of the near-inertial waves becomes comparable to tide itself and dynamics become fully non-linear. The observed decrease flux is approximately 15 percent, much less than was found in previous numerical studies but in reasonable agreement with recent estimates from observational data taken near the critical latitude in the Pacific.

2:21PM QH.00008 Nonlinear effects associated with harmonic forcing of the flow in a rotating sphere, ALBAN SAURET, DAVID CÉBRON, IRPHE, France, CYPRÉN MORIZE, IRPHE, France & FAST, France, MICHAEL LE BARS, STÉPHANE DE ZIÈVES, IRPHE, France — Significant asymmetric stationary flows can be generated by the action of an harmonic forcing on a rotating fluid in a sphere. Such a mechanism could be of fundamental importance in natural systems, for instance in planetary cores subject to libration, precession or tides. Using a weakly non-linear analysis, we first show that the mechanism of zonal flow generation is fully generic: the main contribution in the bulk always comes from the non-linear mechanism. A second study presents the zonal flows amplitude and structure as the square value of the considered forcing and is independent of the Ekman number. These results are confirmed by systematic PIV measurements in a rotating sphere and by asymmetric simulations for the case of libration.

2:34PM QH.00009 The role of non-QG dynamics in the formation and equilibration of eddies in the ocean and atmosphere, J.R. TAYLOR, Massachusetts Institute of Technology, K.S. SMITH, New York University, R. FERRARI, Massachusetts Institute of Technology — The goal of this research is to examine the effect of unbalanced dynamics on the generation and evolution of eddies in the ocean and atmosphere. Numerical simulations of an eddy field generated through baroclinic instability of a jet in geostrophic balance on an f-plane will be presented. Two numerical codes are used, one solves the Boussinesq nonhydrostatic Navier-Stokes equations, and the other solves the quasi-geostrophic (QG) equations. Consistent with previous literature, loss of balance arises spontaneously from the balanced initial condition in the Boussinesq model, but not in the QG model which filters out any inertia-gravity and high Rossby number motions. The novelty of our approach is that we can investigate the role of unbalanced motions on the evolution of the eddy field by considering simulations run with the two models. Pairs of simulations starting at progressively larger Rossby numbers show that non-QG dynamics become more important as the Rossby number is increased. Particular emphasis will be placed on the forward cascade of tracer variance, enstrophy, and energy from large to small scales.

Tuesday, November 23, 2010 12:50PM - 2:47PM — Session QJ Flow Control VI — Long Beach Convention Center 201A

12:50PM QJ.00001 Numerical and Experimental Investigation of Plasma Actuator Control of Modified Flat-back Airfoil, BENJAMIN MERTZ, THOMAS CORKE, University of Notre Dame — Flat-back airfoil designs have been proposed for use on the inboard portion of large wind turbine blades because of their good structural characteristics. These structural characteristics are achieved by adding material to the aft portion of the airfoil while maintaining the camber of the original airfoil shape. The result is a flat vertical trailing edge which increases the overall trends were best modeled through a pressure dependence of the Debye length.

1:03PM QJ.00002 Airfoil Leading Edge Flow Separation Control using DBD Plasma Actuators driven by Nanosecond Pulses1, JESSE LITTLE, University of Arizona, CHRIS RETHMEL, KEISUKE TAKASHIMA, CHRIS WIET, IGOR ADAMOVICH, MO SAMMY, The Ohio State University — This work continues an ongoing exploration of the use of dielectric barrier discharge plasma actuators driven by repetitive nanosecond pulses (NS-DBD hereafter) for aerodynamic flow control. The NS-DBD transfers very little momentum to the neutral air, but generates compression waves that manipulate flow instabilities similar to localized arc filament plasma actuators. Such devices which are believed to function through thermal effects and instability manipulation could result in a significant improvement over conventional DBD (AC-DBD) plasmas that rely on momentum addition which limits their performance at high speeds. The efficacy of NS-DBDs has been demonstrated in our laboratory in a preliminary work on an airfoil leading edge up to Mach 0.17 and Re=1x106. The current work extends the investigation to higher Mach (0.27) and Re (1.15x106), the maximum operating conditions of our subsonic wind tunnel, using an 8 inch chord NACA 0015 airfoil. Results show the efficacy of the nanosecond pulse plasma discharge for attaching the nominally separated flow at various post stall angles of attack.

1Supported by AFRL

1:16PM QJ.00003 Experimental study and optimization of Plasma Actuators for Flow control in subsonic regime, PRADEEP MOISE, JOSEPH MATHEW, KARTIK VENKATRAMAN, JOY THOMAS, Indian Institute of Science, FLOW CONTROL TEAM — The induced jet produced by a dielectric barrier discharge (DBD) setup is capable of preventing flow separation on airfoils at high angles of attack. The effect of various parameters on the velocity of this induced jet was studied experimentally. The glow discharge was created at atmospheric conditions by using a high voltage RF power supply. Flow visualization, photographic studies of the plasma, and hot-wire measurements on the induced jet were performed. The parametric investigation of the characteristics of the plasma show that the width of the plasma in the uniform glow discharge regime was an indication of the velocity induced. It was observed that the spanwise and streamwise overlap of the two electrodes, dielectric thickness, voltage and frequency of the applied voltage are the major parameters that govern the velocity and the extent of plasma. The effect of the optimized configuration on the performance characteristics of an airfoil was studied experimentally.

1:29PM QJ.00004 Pressure Dependence of Plasma Actuated Flow Control, JOSEPH VALERIOTI, THOMAS CORKE, University of Notre Dame — An experimental investigation was conducted to determine how Single Dielectric-Boundary Discharge (SDBD) plasma actuators performed under variable ambient pressure. The static pressure was varied from 0.17 to 9.0 bar. The plasma initiation voltage and static thrust were measured and compared to similar data in literature. The results showed that at a given pressure, the plasma initiation voltage scaled with the actuator capacitor per unit area. The measured thrust showed the previously observed power-law relation with voltage, but the exponent varied with pressure. These trends were evaluated against simulations from the SDBD Space-Time Lumped Element Model. Parameters in the model affected by ambient pressure (capacitance, resistance, and Debye length of the air) were then systematically investigated to determine their effects on the plasma-produced body force. The overall trends were best modeled through a pressure dependence of the Debye length.
1:42PM QJ.00005 Coherent Structures in a Supersonic Jet Excited by Plasma Actuators, DATTA GAITONDE, Air Force Research Laboratory, MO SAMIMY, The Ohio State University — Simulations are used in conjunction with experimental measurements to understand the coherent structures generated by excitation of a Mach 1.3 jet by eight localized arc filament plasma actuators uniformly distributed just upstream of the nozzle exit. Several modes are excited, including the axisymmetric (m=0), helical (m=1-3), and mixed modes (m=±1, ±2) modes. The Strouhal number for all cases is fixed at 0.3, which corresponds to the most amplified frequency. The simulations reproduce the distinct coherent structures measured in the experiment for each azimuthal mode. Detailed analysis of instantaneous, time- and phase-averaged quantities highlights a complex coherent structure generation, evolution and dissipation process. A key feature observed is the initiation of hairpin-like structures with tips/heads in the outer region of the jet shear layer and legs extending forward and slightly inclined in the direction of the jet axis, where the velocity is higher. The subsequent interactions of these structures yield different composite structures in the downstream region. For example, for m=0, adjacent hairpin structures merge to yield axisymmetric rings, with the legs connecting successive structures in the form of ribs in the braid region; and with m=±1 and 0 mode excitation, distinct helical and double-helical structures are observed, respectively, with the hairpins forming substructures in the coils.

1:55PM QJ.00006 Towards Feedback Control of Bypass Transition: Experiments on Laminar Boundary Layer Response to Dynamically Actuated Roughness1, KYLE BADE, AHMED NAGUIB, Michigan State University, Mechanical Engineering, RONALD HANSON, PHILIPPE LAVOIE, University of Toronto, Institute for Aerospace Studies — The current work details observations of the growth of streamwise streaks emanating from cylindrical roughness elements undergoing dynamic actuation into-and-out of a Blasius boundary layer flow. The growth and streamwise propagation of these motions is of interest in a larger study in collaboration with Princeton University in which a multi-university effort aims to develop and implement a robust feedback control system for the weakening/elimination of the streaks (because of their role in initiating bypass transition). Phase-averaged hotwire measurements in the transverse and spanwise directions provide two-dimensional visualizations of the spatial and temporal growth of these motions. Various roughness heights as well as actuation velocities are examined in order to identify the actuation parameters range for which the streaks can be produced while avoiding the introduction of T-S wave packets. This work validates the ability to introduce the proper disturbances into the boundary layer in preparation for the follow up control study.

2:08PM QJ.00007 Towards Feedback Control of Bypass Transition: Experiments on Laminar Boundary Layer Response to a Pulsed Plasma Actuator1, PHILIPPE LAVOIE, RONALD HANSON, University of Toronto, Institute for Aerospace Studies, AHMED NAGUIB, Michigan State University, Mechanical Engineering — Plasma actuators have recently been shown to negate the effect of the transient growth instability occurring in a Blasius boundary layer for the purpose of delaying bypass transition. Specifically, a spanwise array of symmetric plasma actuators generate a counter disturbance of spanwise periodic counter-rotating vortices. During steady operation, the total disturbance energy, introduced via an array of static cylindrical roughness elements, was reduced by up to 68%, as shown by Hanson et al (Exp. Fluids, 2010). The objective of this work is to elucidate the dynamic response of a laminar boundary layer to pulsed excitation by the actuators used in the aforementioned study. The temporal evolution and decay of the disturbance is studied using phase-averaged hotwire measurements at a single plane located downstream of the actuator. The data provide insight into the spatio-temporal character of the modes excited by pulsed plasma actuation. Results are discussed with respect to eventual integration with a feedback control system in collaboration with Princeton University in a multi-university research program aimed at transition control.

2:21PM QJ.00008 Towards Feedback Control of Bypass Transition: Numerical Simulations of Laminar Boundary Layer Response to a Plasma Actuator1, BRANDT BELSON, CLARENCE ROWLEY, Princeton University — We study the effects of single dielectric barrier discharge (SDBD) plasma actuators as a means to delay bypass transition in the Blasius boundary layer, with the eventual goal of closed-loop control. Since streamwise streaks are the structures with the largest transient growth, we orient an array of plasma actuators so as to produce spanwise forces and streamwise vorticity, and thus directly cancel the streaks. We use a pseudo-spectral solver to perform direct numerical simulations of the effect of plasma actuators, implemented as body forces. We compare two different models for the plasma actuator, and then apply each model to our spanwise geometry. We go on to compare each model’s simulation results with experiments carried out by our collaborators at University of Toronto and Michigan State University as part of a multi-university research project.

2:34PM QJ.00009 Improving Wind Turbine Efficiency with Plasma Actuators, JOHN COONEY, THOMAS CORKE, ROBERT NELSON, University of Notre Dame — As increasing the efficiency of modern wind turbines becomes more difficult, the use of active flow control now represents a more attractive means of possible improvement. This ongoing study examines utilizing single dielectric barrier discharge (SDBD) plasma actuators on wind turbine rotors to increase power generation. Blade element momentum (BEM) theory is used to identify regimes with the greatest potential for improvement and to estimate possible gains. Wind tunnel tests are conducted with plasma actuators to determine the amount of aerodynamic control achievable. In addition, the scope of a new “Laboratory for Enhanced Wind Energy Design” is outlined. Most critically, this resource includes two full-scale wind turbines to balance the known limitations of existing theory and wind tunnel testing by providing the capability to test novel blade designs and control strategies in the field.

Tuesday, November 23, 2010 12:50PM - 2:47PM – Session QK Biofluids: Cellular III Long Beach Convention Center 201B

12:50PM QK.00001 Stochastic Intravasation Model for Cancer Metastasis1, ANGELA LEE, STEPHEN LIAO, PAUL NEWTON, University of Southern California — We develop a two-part model that simulates circulating tumor cells (CTCs) entering and then traveling through the human vasculatory system. The first part of our model explores a three-dimensional cluster of CTCs attached to a blood vessel wall in a linear shear flow. The surface of the cells is represented by a 2D Gaussian probability distribution function, and it is discretized with regularized Stokeslets at each grid point. As the system of cells grows stochastically over time, one or more of the cells can detach from the system when the shear forces on the surface exceed a maximum threshold value. In the second part of our model, the newly free-floating CTCs are treated as a dynamical system of multiple, interacting point particles. These particles are represented by singular Stokeslets that are serially introduced into the flow, and the trajectory of each is calculated. The influence of the blood vessel wall is included using the method of images for Stokeslets for a plane boundary. Additional regularized Stokeslets without images are included to represent ambient white blood cells in the bloodstream.

1Supported by the National Science Foundation, award CMMI-0932928.

2Supported by the National Science Foundation, award CMMI-0932928.

3Supported by the National Science Foundation, award CMMI-0932928.
1:03PM QK.00002 Dynamics of a microsphere in an anisotropic gel: a frontier in intracellular microrheology. MANUEL GOMEZ-GONZALEZ, KATHRYN OSTERDAY, JUAN C. DEL ALAMO, University of California San Diego — Particle tracking microrheology determines the properties of a viscoelastic medium from the measured resistance of a moving, immersed microsphere. A crucial assumption in this method is that the medium is isotropic and the sphere experiences Stokes drag. However, the intracellular domain usually presents a pronounced directional structure and anisotropic rheological properties. Current lack of understanding of the dynamics of the probe in this complex environment challenges the application of microrheology to live cells. To overcome this difficulty, we study the drag force experienced by a microsphere in an anisotropic viscoelastic network (the cytoskeleton) permeated by a background liquid (the cytosol). In the limit of strong frictional coupling between the network and the liquid, the drag force is approximated as a function of the anisotropic properties of the network. For high levels of anisotropy, such as those encountered in live cells, previous methods that assume Stokes drag for different effective viscosities along different directions become ill-posed due to the incompressibility of the background liquid.

1:16PM QK.00003 Probing the directional structure and intracellular microrheology of the eukaryotic cytoplasm. KATHRYN OSTERDAY, MANUEL GOMEZ-GONZALEZ, JULIE LI, GERARD NORTICH, JUAN C. LASHERAS, SHU CHEN, JUAN C. DEL ALAMO, University of California, San Diego — The rheological properties of the cytoplasm of animal cells play an important role in cell functions such as migration, mechanotransduction, etc. The magnitude of these properties is important because it sets the level of intracellular deformation in response to stress. The directionality of these properties is equally important because it allows the cell to modulate the stress-strain relation differently along different directions. We aim to elucidate the relation between the structural organization of the cytoplasm and the directionality of its rheological properties by 1) measuring the local orientation of fluorescently labeled intracellular filaments and 2) determining the local directions of the maximum and minimum intracellular viscosity. Directional intracellular viscosities are measured by tracking the random motion of endogenous particles in 2D and applying novel microrheology formulae obtained by studying the motion of a microsphere in a transversely isotropic fluid. Our results indicate that the local viscosity is lowest along the direction parallel to the filaments and that the viscosity in the perpendicular direction is approximately 5 times larger. Under these conditions previous microrheology methods that assumed Stokes drag for the particles have errors in excess of 500%.

1:29PM QK.00004 Do choanoflagellate cells cooperate hydrodynamically to increase feeding fluxes? MARCUS ROPER, JON WILKENING, Dept. of Mathematics, UC Berkeley, M.A.R. KOEHL, Integrative Biology, UC Berkeley, RACHEL PEPPPER, Dept. of Physics, CU Boulder, MARK DAYEL, NICOLE KING, Molecular and Cellular Biology, UC Berkeley — Salpingoeca rosetta is a choanoflagellate, a protozoan that creates water flow with a single flagellum and captures bacterial prey on a collar of microvilli around its flagellum. In response to certain environmental cues S. rosetta switches between unicellular and colonial forms. Analysis of this transition may provide clues about the evolutionary and physical forces that guide the first emergence of colonial life. Our experiments and numerical models show how colonial living changes the feeding currents generated by cells within the colony. Our models also reveal the hidden potential for conflict among the cells in a colony by allowing direct calculation of the iniquitous division of cooperative benefits and costs between cells.

1:42PM QK.00005 ABSTRACT WITHDRAWN —

1:55PM QK.00006 Cellular flow in a small blood vessel. JONATHAN FREUND, MARA ORESCANIN, University of Illinois at Urbana-Champaign — In the tubes and vessels with diameters \( D < 8 \mu m \) red blood cells organize into single-file trains. Simulations are used to investigate flow in a model blood vessel slightly larger than this, \( D = 11.3 \mu m \), for which the cells deviate from this single-file arrangement, deforming continuously and significantly. The effective viscosity of the flow is found to become shear-rate insensitive at higher shear rates \( (U/D > 50 s^{-1}) \) and to match experimental data. At lower shear rates (down to \( U/D = 3.7 s^{-1} \)), the effective viscosity increases by over 50 percent. The cell-free layer that forms along the vessel walls thickens with increasing shear rate and is the key factor governing the overall flow resistance. Cells near the vessel wall are on average inclined relative to the wall, as might be expected for a lubrication mechanism leading to its formation. Metrics are developed to quantify the kinematics in terms of the well-known tank-treading and tumbling behaviors of isolated cells. These rates are found to scale with the velocity difference across the cell-rich core and are thus significantly slower than the overall shear rate in the flow.

2:08PM QK.00007 Cell Transport in Microchannel. A.T. CONILISK, ZHIZI PENG, DANIEL HOYING, The Ohio State University — Cell transport through microscale channels occurs in many biomedical applications such as cell separation by magnetic/electromagnetic forces and cell injection in flow cytometry. Few studies have been performed to understand the motion of the cells as they travel through a microfluidic channel. The objective of this project is to model the velocity of the cells passing through a microfluidic channel under the action of both pressure driven and electrically driven flow fields. Two candidate models of cell transport will be considered. First, the cell transport will be modeled by considering it carried by Poiseuille or Couette flow. Our experiments and numerical models show how colonial living changes the feeding currents generated by cells within the colony. Our models also reveal the hidden potential for conflict among the cells in a colony by allowing direct calculation of the iniquitous division of cooperative benefits and costs between cells.

2:21PM QK.00008 Sticks in honey - Motor-connected Microtubules at low Reynolds number. ROTEM S. BERMAN, Department of Physics, Technion -IIT, ALEX LESHANSKY, Department of Chemical Engineering, Technion -IIT, JOSEPH E. AVRON, Department of Physics, Technion -IIT — It is known that suspensions of microtubules (MTs) and molecular motors spontaneously form ordered arrays. We consider the motion of MTs’ assemblages connected by molecular motors at low Reynolds number. The MTs are modeled as rigid sticks and their hydrodynamic interaction with the medium is determined using slender body approximation. The motors are modeled as moving points which provide kinematic constrains for the sticks’ motion. The hydrodynamic alignment of a pair of MTs for two possible motor connections is considered: a single head motor connection, in which the motor moves on one of the sticks and carries the other one, and a dual head motor connection whereas the motor advances on both sticks. Our experiments and numerical models show how colonial living changes the feeding currents generated by cells within the colony. Our models also reveal the hidden potential for conflict among the cells in a colony by allowing direct calculation of the iniquitous division of cooperative benefits and costs between cells.

2:34PM QK.00009 Time course of pH change in plant epidermis using microscopic pH imaging system. RISAKO DAN, International Christian University, MEGUMI SHIMIZU, Duke University, HARUKO KAZAMA, International Christian University, HIROTAKA SAKAUE, Japan Aerospace Exploration Agency — We established a microscopic pH imaging system to track the time course of pH change in plant epidermis in vivo. In the previous research, we have found out that anthocyanin containing cells have higher pH. However, it was not clear whether the anthocyanin increased the pH or anthocyanin was synthesized result from the higher pH. Therefore, we further investigated the relationship between anthocyanin and pH change. To track the time course of pH change in plant epidermis, we established a system using luminescent imaging technique. We used HPTS (8-Hydroxypterylene-1.3.6-Trisulfonate) as pH indicator and applied excitation ratio imaging method. Luminescent image was converted to a pH distribution by obtained in vitro calibration using known pH solution. Cellular level observation was enabled by merging microscopic color picture of the same region to the pH change image. The established system was applied to epidermal cells of red-tip leaf lettuce, Lactuca Sativa L. and the time course was tracked in the growth process. We would discuss about the relationship between anthocyanin and pH change in plant epidermis.
8:30AM QL.00001 Quantification of hepatic flow distribution using particle tracking for patient specific virtual Fontan surgery
WESTGUANG YANG, UCSD, IRENE VIGNON-CLEMENTE, INRIA France, GUILLAUME TROJANOWSKI, Stanford University, SHAWN SHADDEN, Illinois Institute of Technology, V. MOHAN REDDY, JEFFREY FEINSTEIN, Stanford University, ALISON MARSDEN, UCSD — The Fontan surgery is a third and final stage in a palliative series to treat children with single ventricle heart defects. In the extracardiac Fontan procedure, the inferior vena cava (IVC) is connected to the pulmonary arteries via a tube-shaped Gore-tex graft. Clinical observations have shown that the absence of a hepatic factor, carried in the IVC flow, can cause pulmonary arteriovenous malformations. Although it is clear that hepatic flow distribution is an important determinant of Fontan performance, few studies have quantified its relation to Fontan design. In this study, we virtually implanted three types of grafts (T-junction, offset and Y-graft) into 5 patient specific models of the Glenn (stage 2) anatomy. We then performed 3D time-dependent simulations and systematically compared the IVC flow distribution, energy loss, and pressure levels in different surgical designs. A robustness test is performed to evaluate the sensitivity of hepatic distribution to pulmonary flow split. Results show that the Y-graft design effectively improves the IVC flow distribution, compared to traditional designs and that surgical designs could be customized on a patient-by-patient basis. 

1:03PM QL.00002 Impact of surgical shape on blood flow pattern for patient specific coronary artery bypass graft (CABG) surgery
SETHURAMAN SANKARAN, Postdoctoral fellow, UCSD, ALISON MARSDEN, Assistant Professor, UCSD — We present a numerical framework for studying blood flow patterns in patients who have undergone coronary artery bypass surgeries. We use a stabilized finite element framework for performing blood flow simulations. Specialized lumped parameter boundary conditions for the coronary arteries, aorta and its branches are utilized. Computational models of CABG patients are constructed from CT scan images. A comprehensive study of how surgical shape affects hemodynamics is presented. Patient-specific CABG surgery has not been performed till date. The objective of this work is to study the effect of surgical geometry on blood flow pattern, especially downstream and in the proximity of the suture locations of the bypass graft. Quantities such as energy efficiency, wall shear stresses and its gradients and oscillatory shear index are extracted and compared for different surgical shapes in a systematic fashion. A framework and results for robust optimization of bypass graft anastomoses in unsteady flow will be presented. Implications of surgical geometry on graft patency will be discussed.

1:16PM QL.00003 Physiologic Simulation of the Fontan Surgery with Variable Wall Properties and Respiration
CHRISTOPHER LONG, YURI BAZILEVS, UCSD, JEFFREY FEINSTEIN, Stanford University, ALISON MARSDEN, UCSD — Children born with single ventricle heart defects typically undergo a surgical procedure known as a total cavopulmonary connection (TCPC). The goal of this work is to perform hemodynamic simulations accounting for motion of the arterial walls in the TCPC. We perform fluid structure interactions (FSI) simulations using an Arbitrary Lagrangian Eulerian (ALE) finite element framework into a patient-specific model of the TCPC. The patient’s post-op anatomy is reconstructed from MRI data. Respiration rate, heart rate, and venous pressures are obtained from catheterization data, and flowrates are obtained from phase contrast MRI data and are used together with a respiratory model. Lumped parameter (RCR) boundary conditions are used at the outlets. This study is the first to introduce variable elastic properties for the different areas of the TCPC, including a Gore-Tex conduit. Quantities such as wall shear stresses and pressures at critical junctions are extracted from the simulation and are compared with pressure tracings from clinical data as well as with rigid wall simulations.

1:29PM QL.00004 Computational Modeling of Blood Flow and Valve Dynamics in Hearts with Hypertrophic Cardiomyopathy
XUDONG ZHENG, RAJAT MITTAL, THEODORE ABRAHAM, AURELIO PINHEIRO, The Johns Hopkins University — Hypertrophic cardiomyopathy (HCM) is a cardiovascular disease manifested by the thickening of the ventricular wall and often leads to a partial obstruction of the blood flow out of the left ventricle. HCM is recognized as one of the most common causes of sudden cardiac death in athletes. In a heart with HCM, the hypertrophy usually narrows the blood flow pathway to the aorta and produces a low pressure zone between the mitral valve and the hypertrophy during systole. This low pressure can suck the mitral valve leaflet back and completely block the blood flow into the aorta. In the current study, we virtually implanted three types of grafts (T-junction, offset and Y-graft) into 5 patient specific models of the Glenn (stage 2) anatomy. We then performed 3D time-dependent simulations and systematically compared the IVC flow distribution, energy loss, and pressure levels in different surgical designs. A robustness test is performed to evaluate the sensitivity of hepatic distribution to pulmonary flow split. Results show that the Y-graft design effectively improves the IVC flow distribution, compared to traditional designs and that surgical designs could be customized on a patient-by-patient basis.

1:42PM QL.00005 A Quantitative Study of Simulated Bicuspid Aortic Valves
KAI SZETO, TRAN NGUYEN, University of California, San Diego, JAVIER RODRIGUEZ, Universidad Carlos III, Madrid, Spain, PETER PASTUSZKO, University of California, San Diego, Rady Children's Hospital, San Diego, VISHAL NIGAM, University of California, San Diego, Children's Hospital, San Diego, JUAN LASHERAS, University of California, San Diego — Previous studies have shown that congenitally bicuspid aortic valves develop degenerative diseases earlier than the standard trileaflet, but the causes are not well understood. It has been hypothesized that the asymmetrical flow patterns and turbulence found in the bileaflet valves together with abnormally high levels of strain may result in an early thickening and eventually calcification and stenosis. Central to this hypothesis is the need for a precise quantification of the differences in the strain rate levels between bileaflet and trileaflet valves. We present here some in-vitro dynamic measurements of the spatial variation of the strain rate in pig aortic valves conducted in a left ventricular heart flow simulator device. We measure the strain rate of each leaflet during the whole cardiac cycle using phase-locked stereoscopic three-dimensional image surface reconstruction techniques. The bicuspid case is simulated by surgically stitching two of the leaflets in a normal valve.

1:55PM QL.00006 Particle Image Velocimetry studies of bicuspid aortic valve hemodynamics
NEELAKANTAN SAIRIKRISHNAN, CHOONG-HWAI YAP, AJIT P. YOGANATHAN, Georgia Institute of Technology — Bicuspid aortic valves (BAVs) are a congenital anomaly of the aortic valve with two fused leaflets, affecting about 1-2% of the population. BAV patients have much higher incidence of valve calcification & aortic dilatation, which may be related to altered mechanical forces from BAV hemodynamics. This study aims to characterize BAV hemodynamics using Particle Image Velocimetry (PIV). BAV models are constructed from normal explanted porcine aortic valves by suturing two leaflets together. The valves are mounted in an acrylic chamber with two sinuses & tested in a pulsatile flow loop at physiological conditions. 2D PIV is performed to obtain flow fields in three planes downstream of the valve. The stenosed BAV causes an eccentric jet, resulting in a very strong vortex in the normal sinus. The bicuspid sinus vortex appears much weaker, but more unstable. Unsteady oscillatory shear stresses are also observed, which have been associated with adverse biological response; characterization of the hemodynamics of BAVs will provide the first step to understanding these processes better. Results from multiple BAV models of varying levels of stenosis will be presented & higher stenosis corresponded to stronger jets & increased aortic wall shear stresses.

1:55PM QL.00006 Particle Image Velocimetry studies of bicuspid aortic valve hemodynamics
NEELAKANTAN SAIRIKRISHNAN, CHOONG-HWAI YAP, AJIT P. YOGANATHAN, Georgia Institute of Technology — Bicuspid aortic valves (BAVs) are a congenital anomaly of the aortic valve with two fused leaflets, affecting about 1-2% of the population. BAV patients have much higher incidence of valve calcification & aortic dilatation, which may be related to altered mechanical forces from BAV hemodynamics. This study aims to characterize BAV hemodynamics using Particle Image Velocimetry (PIV). BAV models are constructed from normal explanted porcine aortic valves by suturing two leaflets together. The valves are mounted in an acrylic chamber with two sinuses & tested in a pulsatile flow loop at physiological conditions. 2D PIV is performed to obtain flow fields in three planes downstream of the valve. The stenosed BAV causes an eccentric jet, resulting in a very strong vortex in the normal sinus. The bicuspid sinus vortex appears much weaker, but more unstable. Unsteady oscillatory shear stresses are also observed, which have been associated with adverse biological response; characterization of the hemodynamics of BAVs will provide the first step to understanding these processes better. Results from multiple BAV models of varying levels of stenosis will be presented & higher stenosis corresponded to stronger jets & increased aortic wall shear stresses.

1National Institutes of Health, American Heart Association
2:08PM QL.00007 The Effects of Magnetic Nanoparticles on Magnetic Fluid Hyperthermia

MONRUEE LIANGRUKSA, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, 24061, USA; RAVI KAPPIYOD, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, 24061, USA; RAJAN GANGULY, Department of Power Engineering, Jadavpur University, Kolkata 700032, India; ISHWAR PURI, Department of Power Engineering, Jadavpur University, Kolkata 700032, India; ISHWAR PURI, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, 24061, USA; HELENE SIMON, Georgia Institute of Technology — Magnetic fluid hyperthermia (MFH) is a cancer treatment in which biocompatible magnetic nanoparticles are dispersed into a tumor and heated by an external AC magnetic field. Over a period of time, the tumor cells are locally heated, leading to hyperthermia which damages and kills the tumor cells with minimal damage to the surrounding normal tissue. The applied magnetic field must be high enough to induce hyperthermia for a specified magnetic particle concentration in the tumor but also lies within the safe limit for human exposure. Six materials, barium ferrite, cobalt ferrite, iron-cobalt, iron-platinum, magnetite and maghemite, are considered as candidates for MFH use. We find that fcc iron-platinum, magnetite and maghemite generate useful treatment temperatures and, when included in a ferrofluid, can produce sufficient and effective MFH for which optimal conditions are explored.

2:21PM QL.00008 A numerical investigation of blood damage in the hinge area of bileaflet mechanical heart valves

MIN YUN, JINGSHU WU, HELENE SIMON, Georgia Institute of Technology, FOTIS SOTIROPOULOS, University of Minnesota, CYRUS AIDUN, AJIT YOGANATHAN, Georgia Institute of Technology — Studies have shown that high shear stress and large recirculation regions have a strong impact on thromboembolic complications in Bileaflet mechanical heart valves (BMHV). This study quantitatively compares the hinge flow field and blood damage of the 23mm St. Jude Medical (SJM) regent with different hinge gap widths and the 23mm CarboMedics (CM) valves. The lattice-Boltzmann method with external boundary force (LBM-EBF) [Wu and Aidun, Int. J Num. Methods Fluids, 62, 7, 2009] was implemented to simulate the flow and capture the dynamics and the surface shear stress of the platelets with realistic geometry. The velocity boundary conditions for the small-scale hinge flow are obtained from previous 3D large-scale computational fluid dynamics (CFD) simulations [Simon et al., Annals of Biomedical Engineering, 38, 3, 2009]. The flow patterns of three hinges that were studied were similar during diastole. However, velocity magnitudes and shear stresses at the hinge gap were different, which may explain the higher blood damage index (BDI) value for the CM valve and lower BDI value for the SJM valve with a larger gap width. The multiscale computational method used to quantitatively measure the BDI during a full cardiac cycle will be discussed.

Tuesday, November 23, 2010 12:50PM - 2:47PM –
Session QM Free Surface Flows: Droplets and Sheets Long Beach Convention Center 202B

12:50PM QM.00001 Capillary retraction of liquid sheet

GILLES AGBAGLAH, CHRISTOPHE JOSERAND, STÉPHANE ZALESKI, Institut d'Alembert, Paris. CNRS & UPMC — During the atomization, drops may be formed by several distinct mechanisms. A general understanding of these processes is still lacking and is at the heart of many fundamental studies on atomization. In particular, the destabilization of a liquid sheet is known to detach small droplets. In this work, retracting liquid sheet is numerically studied in 2D and 3D. We present an asymptotic expansion of the film profile in 2D and we develop the long wave approximation dynamics of a planar 3D sheets. The role played by the ambient gas and new instabilities for the retracting liquid sheet is also discussed.

1:03PM QM.00002 Atomization patterns of liquid sheets formed by two impinging jets

DON-JUN MA, XIAO-DONG CHEN, VIGOR YANG, School of Aerospace Engineering, Georgia Institute of Technology — High fidelity numerical simulations have been performed to study the atomization patterns and breakup characteristics of liquid sheets formed by two impinging jets. A fully three-dimensional Volume-of-Fluid method with adaptive mesh refinement (AMR) based on octree-mesh is used to simulate the primary atomization. The start of the art visualization techniques with volume rendering were also used to highlight the breakup characteristics. The oblique collision of two cylindrical laminar jets leads to the liquid owing outward from the impact point, creating a thin sheet which lies in a plane perpendicular to the plane containing the two jets and disintegrates into ligaments or droplets. The breakup of the sheet is dominant by the viscosity and surface tension effects (Reynolds and Webber number). The periodic waves from the point of impingement were apparent on the surface of the sheet. The circumferentially space drops were shed from the periphery of the sheet, as well as the ligaments were fragmented from the leading edge of the sheet and then broke into droplets following the Rayleigh mechanism. The impact waves caused early breakdown of the sheet downstream of the impingement point, whereas waves amplified by aerodynamic stresses controlled the breakdown of the rest of the sheet and the ligaments.

1:16PM QM.00003 Vibration-Induced Gas-Liquid Interface Breakup

TIMOTHY O’HERN, JOHN TORCZYNSKI, ED ROMERO, BION SHELDER, Sandia National Laboratories — Gas-liquid interfaces can be forced to break up when subjected to vibrations within critical ranges of frequency and amplitude. This breakup mechanism was examined experimentally using deep layers of silicone oils over a range of viscosity and sinusoidal, primarily axial vibration conditions that can produce dramatic disturbances at the gas-liquid free surface. Although small-amplitude vibrations produce standing Faraday waves, large-amplitude vibrations produce liquid jets into the gas, droplets pinching off from the jets, gas cavities in the liquid from droplet impact, and bubble transport below the interface. Experiments used several different silicone oils over a range of pressures and vibration conditions. Computational simulations exhibiting similar behavior will be included in the presentation. Applications include liquid fuel rockets, inertial sensing devices, moving vehicles, mixing processes, and acoustic excitation. Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

1:29PM QM.00004 Crumpled liquid sheet

HENRI LHUSSIERS, EMMANUEL VILLOMAUX, Aix-Marseille Universite, IRPHE — When a liquid jet of density ρ impacts a solid disk at right angle, it expands radially into a thin sheet with velocity u and thickness h. The sheet possibly bends under the action of surface tension σ and supercritical (We ≥ Wc_s) and subcritical (We < Wc_s) regions downstream.
We propose several scaling arguments for the formation of this important household phenomenon.

2:08PM QM.00007 The Tibetan singing bowl, JOHN BUSH, MIT, DENIS TERWAGNE, University of Liege — Tibetan singing bowls have been used for centuries for healing, meditation and shamanic journeying. The bowls are partially filled with water, then excited by either striking or rubbing the walls of the bowl with a mallet. A wealth of curious fluid mechanical phenomena arise, and will be elucidated in our combined experimental and theoretical investigation.

2:21PM QM.00008 Inviscid Breakup of Bubbles and Drops With and Without Surface Charge, JUSTIN BURTON, University of Chicago, PETER TABOREK, University of California, Irvine — We present boundary-integral simulations of the breakup of inviscid bubbles and droplets, with and without surface charge. In our simulations, an inner fluid volume of density ρ1 is surrounded by an exterior fluid of infinite extent and density ρ2. When there is no charge on the surface, we see excellent agreement with previous work, except for intermediate density ratios, where the simulations are plagued by oscillatory instabilities not observed in experiments [1]. With the addition of surface charge, initially spherical droplets and bubbles are unstable to small perturbations above a critical surface charge density. For the droplet limit, the charged drop forms a “lemon” shape before ejecting a highly charged jet from the tips of the “lemon,” where the size of the jet scales with the square of the inverse surface conductivity. For the bubble limit, we find that fission always takes place by the formation of a “peanut”-shaped bubble, where breakup takes place at the center of the bubble, regardless of surface conductivity. For intermediate densities, combinations of droplet and bubble fission are observed.


Tuesday, November 23, 2010 12:50PM - 2:47PM — Session QN Granular Flows III Long Beach Convention Center 202C

12:50PM QN.00001 Granular materials: on topology of force chains, LOU KONDIC, MICHEL TSUKAHARA, NJIT, MIROSLAV KRAMAR, KONSTANTIN MISCHAIKOW, Rutgers University, JIE ZHANG, ROBERT BEHRINGER, Duke University, COREY O’HERN, Yale University — One property of granular materials is the lack of spatial scale separation between the one characterizing the particle size and the one characterizing system as a whole. This property requires careful understanding of the features which exist on particle scale, such as force chains, with the hope that this understanding will help us produce appropriate continuum level models. In this talk, we will discuss our initial attempts to characterize force chains. These attempts are based on algebraic topology techniques that will be used to analyze and quantify force chain structures. In particular, we will discuss how these properties differ for the systems exposed to shear versus compression, and correlate the topological measures to the phenomena such as jamming. Furthermore, we will discuss the possibility of using topological techniques to come up with a quantitative way of comparing experiments and simulations. Preliminary results of these comparisons will be shown.

1 Supported by NSF Grant No. DMS-0835611.

1:03PM QN.00002 Energy bursts in shallow granular systems, NICOLÁS MUJICA, NICOLÁS RIVAS, SUOMI PONCE, Departamento de Fisica, FCFM, Universidad de Chile, BASILE GALLET, Laboratoire de Physique Statistique, Ecole Normale Superieure, DINO RISSO, Departamento de Fisica, Universidad del Bio-Bio, RODRIGO SOTO, PATRICIO CORDERO, Departamento de Fisica, FCFM, Universidad de Chile — In a mixture of two species of grains of equal size but differing by their mass, place in a vertically vibrated shallow box, there is spontaneous segregation. Once the system is at least partly segregated, energy bursts take place: the horizontal kinetic energy of the heavy particles, that normally is small, suddenly increases. An explanation is provided based on the existence of a fixed point for an isolated particle bouncing with only vertical motion between the top and bottom walls. Energy bursts occur when the large energy stored in the vertical motion is partly converted into horizontal energy through a chain reaction of collisions between heavy particles. Depending on the experimental or numerical parameters and initial conditions, the energy bursts can occur either randomly or rather periodically in time.

1 Fondecyt Grants No. 1061112, No. 1070958, No. 1090188, No. 1100100, and ECOS C07E07.
1:16PM QN.00003 3D aspects of mixing and transport in tumbled granular flow. IVAN C. CHRISTOV, RICHARD M. LUEPTOW, JULIO M. OTTINO, Northwestern University. ROB STURMAN, University of Leeds. STEPHEN WIGGINS, University of Bristol — The exploration of the kinematic structures that emerge in 3D flows has only just begun (see, e.g., Focus on Fluids in JFM vol. 654, 2010). Tumbled granular flow in a spherical container rotated sequentially about two distinct axes is a convenient physical system in which to investigate these issues. The flow can “switch” between 2D motion (dynamics restricted to 2D manifolds) and fully-3D motion depending on the choice of angles of rotation about the axes. We compute explicitly the action- action-angle transformation, period along trajectories and exact location of normally- hyperbolic and elliptic period-one curves from the piecewise-defined nonlinear dynamical system. This provides the basis for a definition of a 3D notion of an “island.” These theoretical results also allow for the “optimization” of the angles of rotation in the protocol. An extensive numerical investigation of the “goodness of mixing” is performed using Danckwerts’ intensity of segregation $I$. By fitting the decay rate and asymptotic value of $I$, we can understand the effects of the protocol parameters and how mixing varies across the volume of the tumbler. Finally, we establish the existence of “adiabatic” structures (2D manifolds exhibiting chaotic and ergodic dynamics) and study the persistence of these barriers to radial transport as the flow is perturbed into the fully-3D regime.

1:29PM QN.00004 Derivation and numerical treatment of the low-Mach number equations for two-phase granular mixtures. CHRISTOS VARSAKELIS, MILTIADIS PAPALEXANDRIS, Universite catholique de Louvain — In this talk we present a methodology for the derivation of the low-Mach number equations for two-phase flows and apply it to a particular constitutive model for immiscible mixtures of a granular material and a fluid. The proposed methodology is based on non-dimensionalizing the governing equations of the two-phase flow in the inertial and quasistatic regimes and the method of matched asymptotic expansions. The resulting system of equations for the low-Mach number limit of the two-phase flow is then compared with existing results in the literature.

1:42PM QN.00005 Simulations of 2D granular jet impact deadzone formation. NICHOLAS GUTTENBERG, WENDY ZHANG, University of Chicago — Motivated by granular experiments showing the emergence of continuum-like dynamics when a dense jet hits a target, we simulate the impact of a 2D granular jet of frictional, cohesionless grains upon a fixed target. This is an inertial, dense jet regime where the motion is essentially incompressible. Impact deflects the material in the jet into a hollow conical sheet. The cone angles measured in simulation are consistent with previous experimental studies of the 3D granular jet impact. In addition, experiments have revealed the formation of a “dead zone,” a region where the grain motion is negligibly small. The simulation shows that this dead zone can only form when a no-slip boundary condition is enforced at the target. The presence or absence of the dead zone leads to a change in cone angle consistent with the experimentally observed differences in cone angle between the 3D granular flow and the corresponding water bell flow.

1:55PM QN.00006 ABSTRACT WITHDRAWN —

2:08PM QN.00007 Rheology of simple shear flows of dense granular assemblies in different regimes. SEBASTIAN CHIALVO, JIN SUN, SANKARAN SUNDARESAN, Princeton University — Using the discrete element method, simulations of simple shear flow of dense assemblies of frictional particles have been carried out over a range of shear rates and volume fractions in order to characterize the transition from quasistatic or inertial flow to intermediate flow. In agreement with previous results for frictionless spheres [1], the pressure and shear stress in the intermediate regime are found to approach asymptotic power law relations with shear rate; curiously, these asymptotes appear to be common to all intermediate flows regardless of the value of the particle friction coefficient. The scaling relations for stress for the inertial and quasistatic regimes are consistent with a reference thermodynamic state of the phase with the smallest speed of sound. Further, we propose an algorithm for the numerical treatment of these equations. It belongs to the class of fractional-step algorithms and employs a generalized projection method for the momentum equation of each phase. Our discussion concludes with the presentation of some preliminary numerical results for constant density flows.

2:21PM QN.00008 Rheology of Granular Mixtures Differing in Size and/or Density. ANURAG TRIPATHI, D.V. KHAKHAR, Indian Institute of Technology Bombay — Rheology of mono-dispersed granular materials is well understood and it is well known that these materials follow a friction law where the shear stress to pressure ratio is determined by the Inertial number $I$. However, rheology of the general and more common case of granular mixtures of different size and/or different density particles has not received attention of researchers. We study the rheology of binary mixtures flowing over an inclined plane under the influence of gravity by means of DEM simulations. We show that the friction law for single component granular material with appropriate modification in the inertial number expression captures the rheology of the mixtures as well and can predict the viscosity of both, same-size different-density particle mixtures and different-size same-density particle mixtures. For the case of mixture of particles differing in size and density both, we obtain a well-mixed or a segregated state depending upon the over-all composition of the mixture. The modified friction law is able to predict the viscosity for this case as well for both well-mixed and segregated state. Thus we show that friction law with a generalized definition of Inertial number can predict the rheology of granular mixtures differing in size and/or density.

2:34PM QN.00009 Rheology of dense granular mixtures: Particle size distributions, boundary conditions, and collisional time scales. KIMBERLY HILL, BEREKET YOHANNES, SAFL, Department of Civil Engineering, University of Minnesota — We computationally investigate the dependence of the rheology of dense sheared granular mixtures on their particle size distribution. We find that the variation of the rheology with the particle size distribution depends on the boundary conditions. For example, under constant pressure conditions the effective friction coefficient $\mu^*$ (the ratio between shear and pressure stresses at the boundary) increases mildly with the average particle size. On the other hand, under constant volume conditions, $\mu^*$ has a non-monotonic dependence on the average particle size that is related to the proximity of the system solids fraction to the maximum packing fraction. Somewhat surprisingly, then, $\mu^*$ scales with a dimensionless shear rate (a generalized inertial number) in the same way for either boundary condition. We show that, for our system of relatively large spheres, these relationships are governed largely by the ratio between average collision times and mean free path times, also independent of boundary conditions.

Tuesday, November 23, 2010 12:50PM - 3:00PM – Session QP Microfluidics: Mixing II Long Beach Convention Center 203A
12:50PM QP.00001 Optimal advective mixing by two-dimensional chaotic Stokes flows. DAVID SAINTTILLAN, QIZHENG YAN, MechSE, University of Illinois at Urbana-Champaign — Numerous mixing strategies in microfluidic devices rely on chaotic advection by time-dependent body forces. The question of determining the required forcing function to achieve optimal mixing at a given kinetic energy or power input remains however open. Using finite-horizon optimal control theory, we numerically determine general optimal mixing flows in a two-dimensional periodic geometry as truncated sums of time-modulated Fourier modes. The time-averaged power spectra of these flows are calculated to investigate the effect of scale. We demonstrate that optimal mixing flows with fixed kinetic energy contain a wide range of spatial scales, whereas those with fixed power input are strongly dominated by large scales. We also determine the frequency spectra of the time-modulating functions and characterize the importance of non-harmonic forcing.

1:03PM QP.00002 Analysis of Mixing in a 2D Drop with Time-Periodic Boundary Forcing. MICHAEL DAVIS, Claremont Graduate University, AMANDA CLEMM, Scripps College, CECILY KEPEL, DYLAN MARRINER, ANDREW BERNOFF, Harvey Mudd College, ALI NADIM, Claremont Graduate University — We carry out a detailed analysis on the model problem of Nadim & Miraghaie [Bull. Am. Phys. Soc., 49, 188 (2004)], which consists of a 2D circular drop driven by a tangential stress applied at its boundary giving rise to a pair of circulating flows in each half of the drop that are periodically reoriented. We characterize the resulting chaotic flow by computing the Lyapunov Exponents (LE) and their Finite-Time counterparts (FTLE) for all initial positions within the drop. We calculate the mean and variance of the FTLEs for a wide range of switching times, and identify the optimal switching time for efficient mixing. At some non-optimal switching times, the drop domain contains a mixing region and non-mixing islands which are associated with a bimodal distribution of FTLEs. For a certain switching time, we identify a group of 4 points (which form a square in the drop) that are permitted by the flow and return to their original positions after 4 switching periods. The space-time trajectories of these points, which can be regarded as virtual "stirring rods," form braids when the flow is chaotically mixing. Calculation of braiding factors associated with different patterns of switching enables us to assess their mixing efficacy. [Supported by Fletcher Jones Fellowships/CCMS]

1:16PM QP.00003 Low Reynolds number flow over slanted grooves in a micro-channel1. SUNGCHAN YUN, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH), ELECTROMICROFLUIDICS LABORATORY TEAM — A prestress driven flow over a micro-patterned geometry generates a transverse velocity component to the principal direction of flow. In the analysis of the flow inside a micro-channel, for simplicity of analysis, an effective slip velocity is applied to represent the transverse velocity developed by the slanted grooves. However, since the slip model is based on a periodically placed infinite length of linear grooves, the validity of model is limited only for shallow-depth grooves or thin channels. In this work, we investigated flow patterns near grooves in a closed channel based on three-dimensional numerical analysis, and the numerical results are verified experimentally. The fluid flow visualization reveals that the flow pattern becomes somewhat complicated, as the depth or width of the grooves is increased, which cannot be accounted for by the simple slip model. Based on the numerical results, we determined the range of depth and width of the grooves in which the effective slip model can be applied.

1:29PM QP.00004 Designing a Robust Micromixer Based on Fluid Stretching1. DAVID MOTT, Naval Research Laboratory, DIPESH GAUTAM, GREG VOTH, Wesleyan University, ELAINE ORAN, Naval Research Laboratory — A metric for measuring fluid stretching based on finite-time Lyapunov exponents is described, and the use of this metric for optimizing mixing in microfluidic components is explored. The metric is implemented within an automated design approach called the Computational Toolbox (CTB). The CTB designs components by adding geometric features, such as grooves of various shapes, to a microchannel. The transport produced by each of these features in isolation was pre-computed and stored as an “advection map” for that feature, and the flow through a composite geometry that combines these features is calculated rapidly by applying the corresponding maps in sequence. A genetic algorithm search then chooses the feature combination that optimizes a user-specified metric. Metrics based on the variance of concentration were generated by varying the density of microchannels and their orientation. Metrics based on minimizing the generation of chaotic mixing by using out-of-phase pulsing in two-inlet channels, such as T or Y channels. Cases ranging from in-phase to anti-phase inlet pulsing have been simulated to optimize the size of the chaotic mixing region. Poincaré analysis was performed for each of the cases and the area of chaotic mixing was measured to determine the value of the parameters leading to the best mixing results. In addition, Lagrangian Coherent Structures (LCS) were be used to determine the chaotic stretching. The use of this metric for optimizing mixing in microfluidic components is explored. The metrics are implemented within an automated design approach called the Computational Toolbox (CTB).

1:42PM QP.00005 A Two-Equation Model For Mixing in Viscous-Fingering Displacements. BIRENDRRA JHA, LUIS CUETO-FELGUEROOSO, RUBEN JUANES, MIT — We study, by means of numerical simulation, the mixing of two fluids of different viscosities in advection-dominated flows in a porous medium. It is well known that when a less viscous fluid displaces a more viscous fluid, the displacement front is unstable and leads to the formation of a pattern known as viscous fingering. We present a high-resolution simulation approach that is stable for arbitrary viscosity ratios, and study mixing under different configurations with viscosity contrasts up to M = 400. We observe, in agreement with lab experiments, that for high-M mixing, the growth of new fingers follows the trace of previous ones. This channeling effect, which is a result of the nonlocal coupling through the pressure field, greatly reduces mixing. A two-equation mixing model using the scalar variance and its dissipation rate is derived from the advection-diffusion equation. It provides a measure of effective diffusivity due to convective and diffusive mixing processes. Our analysis predicts the optimum range of viscosity contrast and Pelet number that maximizes the interfacial area by balancing the number of fingers with their length before diffusive mixing across the sharp interface takes over. Interesting fingering patterns such as channeling and tip-splitting play an important role in this balancing act which makes degree of mixing a non-monotonic function of the viscosity contrast and the Pelet number.

1:55PM QP.00006 Lubrication Analysis of Flow and Mixing in a 3D Translating Sessile Drop. CECILY KEPEL, Harvey Mudd College, AMANDA CLEMM, Scripps College, MICHAEL DAVIS, Claremont Graduate University, DYLAN MARRINER, ANDREW BERNOFF, Harvey Mudd College, ALI NADIM, Claremont Graduate University — We consider the flow within a sessile drop that translates along a surface. The drop height is taken to be small compared to the radius of its wetted base allowing a lubrication approximation. The drop is also assumed to be well mixed within the drop volume. [Supported by Fletcher Jones Fellowships/CCMS]
2:21PM QP.00008 Passive scalar separation using chaotic advection, ANDREW DUGGLEBY, PRADEEP RAO, Texas A&M University, PANKAJ KUMAR, MARK STREMLER, Virginia Tech — Separation of two substances with slightly different diffusivities using chaotic advection is explored for finite Reynolds numbers (up to Re ~ 10) and high average Schmidt numbers ($Sc = (Sc_1 + Sc_2)/2 \approx 10^6$) for a modified lid-driven cavity. In this approach, exponential stretching of material interfaces enhances diffusion and accelerates separation of concentrated molecules having slightly different diffusivities. At low Re the flow can be reversed and the separated molecules extracted. Using the exponential convergence afforded by the use of a 2D Fourier-Chebyshev spectral algorithm for streamfunction-vorticity formulation with passive scalar transport enables accurate tracking of exponential stretching of material interfaces in the flow and capturing of the sharp concentration gradients associated with large $Sc$. The two substances separate significantly faster than for simple diffusion. Performance based on topological entropy and almost-invariant sets, as well as application to real separation systems, will be discussed.

2:34PM QP.00009 Evidence of streaming-related irreversibility and mixing in low Reynolds number acinar flows¹, HARIBALAN KUMAR, CHING-LONG LIN, Department of Mechanical and Industrial Engineering, The University of Iowa, MERRYN H. TAWHAI, Auckland Bioengineering Institute, The University of Auckland, ERIC A. HOFFMAN, Department of Radiology, Internal Medicine and Biomedical Engineering, The University of Iowa — Understanding kinematic irreversibility and mixing deep in the lung helps improve particle retention estimates and hence provide better drug delivery strategies. The time-periodic low-Reynolds number flow in the tiny alveolar units can be computed using an open cavity configuration. Steady streaming is found to hold the key to the origin of irreversibility and dispersion in the duct, cavity mouth and within the cavity. The mechanism of steady streaming is hydrodynamic in nature. The results of tracer advection and mixing rates are used to quantify the irreversibility and mixing resulting from this streaming. The effect of varying Strouhal numbers, Reynolds numbers and geometrical parameters on the resulting mixing are also investigated. This streaming mechanism may provide a route to explaining dispersion observed in bolus experiments deep in the lung.

1This work was supported in part by NIH Grant numbers R01-HL-094315, R01-EI-005823, R01-HL-064368, and S10-RR-022421.

2:47PM QP.00010 Experimental study on biological mixing by micro-organism, JIHOON KIM, YONGHEE JANG, DOYOUNG BYUN¹, Konkuk University, SONGWON NAM, SUNGSU PARK, Ewha University, MINJUN KIM, Drexel University — Recently, the most challenging in a microfluidic device remains in acting on the device without external source such as syringe pump, magnetic driven force, and electrophoresis driven force. Instead of the artificial external force, biological propelled mechanism has been paid much attention. Most of micro-organisms have shown to generate straight motion, vibration, and rolling motion. Those motions can be applied to numerous part of micro-actuator or biological robot. In this paper, we investigated the flow field induced by swimming Tetrahymena and suggest this for mixing mechanism. Using micro-particle image velocimetry system, we visualized dynamic motions by DC, AC, and AC+DC galvanotaxis. Due to the periodic signal of AC voltage, Tetrahymena swimming is easily controlled on any desired direction. AC galvanotaxis also allows it to stop at a position only by changing the applied frequency and voltage. Therefore, this galvanotactic motion control can be applied to biological micro-mixer in the microfluidic device.

¹Corresponding Author

Tuesday, November 23, 2010 12:50PM - 3:00PM – Session QQ Bubbles IV Long Beach Convention Center 203B

12:50PM QP.00001 Understanding the spontaneous brittle-to-ductile transition in foam fracture, SHEHLA ARIF, Mechanical Engineering, Northwestern University, JIHING-CHIANG TSAI, Institute of Physics, Academia Sinica, Taiwan, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — A single layer of aqueous foam bubbles in a Hele-Shaw cell, when exposed to compressed air, can fail in both a brittle and a ductile fracture mode. Unlike fracture in hard crystalline matter, where either mode can be induced through external parameter changes, we observe that cracks in foam can transition spontaneously from the brittle to the ductile stage. The transition occurs dynamically and is accompanied by the cessation of film rupture and an abrupt change in crack speed obeying a gap of forbidden velocities. The spontaneous transition can be understood through the continuous action of dissipation through air flow in the expanding fracture channel. An accompanying theory based only on fluid dynamics and bubble geometry explains quantitatively the mechanism and location of this transition, as well as the dependence of the phenomenon on experimental parameters. The new insights are applicable beyond foams to physics of films like hydraulic fractures.

1:03PM QP.00002 Deformation of soap films pushed through tubes at high velocity¹, ISABELLE CANTAT, BENJAMIN DOLLET, Rennes University (France) CNRS — The behaviour of soap films pushed through tubes at large velocities, up to several m/s, is investigated. The film shape deviates from its equilibrium configuration perpendicular to the walls and gets curved downstream. A simple model relates the radius of curvature of the film to the friction in the lubrication films touching the wall. For highly soluble surfactants above the cmc, the scaling proposed by Bretherton in 1961 holds up to surprisingly high velocities, at which the capillary and Weber numbers are no longer small parameters. A limit of stability of the film, beyond which the films burst or evolve unsteadily, is predicted, and captures quantitatively the observations. In contrast, an unsteady motion is obtained for insoluble surfactant, governed by strong Marangoni effects. The new questions raised by our results on the dissipation in soap films are discussed and comparison with numerical simulations are proposed.

¹I.C. acknowledges support from the Institut Universitaire de France.

1:16PM QP.00003 The micro-macro link for liquid foam stability¹, ANNE-LAURE BIANCE, LPMCN, Université Lyon 1 et CNRS, ALINE DELBOS, OLIVIER PITOIS, LPMDI, Université Paris-Est Marne La vallée, LPMCN TEAM, LPMDI COLLABORATION — The experimental study of bubble rearrangements triggered artificially in both bubble clusters and liquid foams allows us to establish a link between the stability of the system and the dynamical behaviour of foam films. More precisely, the amount of liquid available locally in the system has to be larger than a critical value to ensure the formation of transient films within dynamical conditions. This has been clearly attributed to a dynamical thickening effect of the fresh film, whose thickness is strongly dependent on the rheological properties of interfaces. A simple model is proposed to capture these new findings and shows good quantitative agreement with measurements for the critical liquid fraction of foam collapse, providing new insight in the understanding of the complex coupling between internal dynamics and stability of foams.

¹The authors would like to thank ESA and CNES for financial support.
1:29PM QQ.00004 Accurate detection of pierced position/angle in bubble measurements by a Single-Tip Optical fiber Probe, YUKI MIZUSHIMA\(^1\), Department of Engineering, Shizuoka University, TAKAYUKI SAITO\(^2\), Graduate School of Sci. and Tech., Shizuoka University — An optical fiber probe (OPF) is very useful and reliable to measure bubble diameters, velocities, and local void fractions simultaneously in bubbly flow systems. One of the authors already developed the Single-Tip Optical-fiber Probe (S-TOP), which is practically employed in small-size-bubble measurement in industrial plants. Its sensing tip is smoothly ground into a wedge shape. In optical fiber probing, errors due to S-TOP tip randomly touching the bubble surface were unavoidable thus far. To overcome this problem, we newly propose a pre-signal method, which is powerful yet simple process. It occurs intensively, only when the well-tuned wedge shape tip touches vertically the center region of the bubble. In this study, at first, we quantify the relationship between the intensity of the pre-signal and the contact position/angle. Second, we discuss the outbreak mechanism of the pre-signal. Third, based on the relationship, a pre-signal method is established. Finally, it is confirmed that the pre-signal method is effective in bubbly flow measurement.

\(^1\)student
\(^2\)professor

1:42PM QQ.00005 Explosive boiling incipience on a thin wire, JEAN-CHARLES Nardin, CÉDRIC POULAIN, CEA Grenoble, JÉRÔME DUPLAT, Université de Provence — When a metastable liquid is superheated above its saturation temperature, a phase transition occurs via a nucleation process leading to the creation of at least one vapor bubble that grows rapidly. If the surrounding liquid is subcooled, the bubble will eventually undergo a violent collapse. A further characterization of the thermodynamic properties of this explosive phase change, (temperature at the onset of nucleation as well as pressure inside the first nucle), together with the following bubble dynamics, is necessary for a better comprehension of boiling phenomena. Thanks to dedicated experiments in which a platinum micrometer–size wire is heated in a liquid at ambient pressure and temperature, we will report that the onset temperature is close to the spinodal temperature but slightly depends on the heating rate. Using high-speed video imaging of the bubble dynamics together with the Rayleigh–Plessset equation, we will show how the heating rate, as well as the heater size governs the nucleation process (bubble lifetime, maximum radius reached, expansion velocity and cooling of the wire at the onset).

1:55PM QQ.00006 Actively Enhanced Boiling Heat Transfer using Acoustic Interfacial Actuation, THOMAS R. BOZIUK, MARC K. SMITH, ARI GLEZER, Georgia Institute of Technology — Acoustic actuation is used to enhance boiling heat transfer on a submerged surface by controlling the formation and evolution of vapor bubbles and inhibiting instabilities that lead to film boiling and critical heat flux. The receptivity of a vapor bubble that forms at a prescribed nucleation site to acoustic interfacial excitation and to acoustically induced Bjerknes body forces is investigated on a surface-embedded hot spot with emphasis on the acoustic effects on nucleation, growth, contact-line motion, condensation, and detachment. The investigation also considers arrays of vapor bubbles that form on a prescribed grid of surface-engineered nucleation sites and the interactions between adjacent vapor bubbles. It is shown that acoustic actuation enables dissipation of higher heat fluxes at a given surface temperature, and a significant delay of the critical heat flux with reduction of the vapor mass above the surface. Supported by ONR.

2:08PM QQ.00007 Infrared Visualisation of Nucleate Boiling from an Isolated Site in a Hele-Shaw Cell, JACQUELINE BARBER, DAVID BRUTIN, LOUNES TADRIST, Polytech Marseille, Université de Provence — Nucleation, growth and detachment of HFE-7000 confined vapour bubbles are investigated locally using simultaneously an infrared camera with 10 \(\mu\)m spatial resolution and a visible camera. Bubbles are created from a 50 \(\mu\)m artificial nucleation site on a 100 \(\mu\)m Inconel film in a Hele-Shaw cell. A single bubble nucleation is investigated. Experiments are performed by varying both the convective inlet liquid mass flow rate, and the heat flux supplied at the wire. Bubble detachment diameters at the single artificial nucleation site and the associated effects on the heat transfer by the confinement influence are investigated at low Reynolds number. The experimental set-up enables observation of local temperature fields and bubble shapes. The interactions between adjacent vapor bubbles is studied. It is shown that the bubble shapes generated are thermally stabilised by the confinement and that the bubble detachment is delayed. The infrared video resolution enables us to observe the presence of a bubble and its magnitude of thermal disturbance on the flow. The temperature profile of the bubble as it nucleates, grows, detaches from the wall can be measured from the infrared videos. The temperature change at the nucleation site highlights the frequency of the bubble detachment based on the temperature signal. From analysis of the visible and infrared videos, the nucleation site surface temperature, bubble detachment diameter and bubble nucleation frequency can be calculated.

2:21PM QQ.00008 Drag force acting on an ellipsoidal bubble with fore-aft asymmetry, TOSHIYUKI SANADA, Shizuoka University, MINORI SHIROKI, Hiroasaki University — We evaluate the drag force acting on an ellipsoidal clean bubbles rising steadily by experiments and numerical analysis. Flow fields and bubble shapes are obtained using a numerical simulation, which is based on a finite-difference solution of the equation of motion on an orthogonal curvilinear coordinate system. Bubble motion in eight different clean liquids are also observed using high-speed photography. Photochromic dye is used to visualize the rear vortex structure. The degree of fore-aft asymmetric bubble shape is quantitatively evaluated using Legendre polynomials. It is confirmed that the existence of standing eddy hardly changes the drag even in the case with fore-aft asymmetry of bubble shape. It is also found that non-dimensional steady drag acting on a bubble has a liner relation with aspect ratio. The discrepancy of drag coefficients between analytical theory and experimental results is discussed.

2:34PM QQ.00009 Conditions for microthread formation in viscous coflows, ELENA DE CASTRO HERNANDEZ, JOSE MANUEL GORDILLO, Grupo de Mecanica de Fluidos, ESI, Universidad de Sevilla, SPAIN — We have performed numerous experiments with coaxial coflowing fluids in microfluidic devices at low Reynolds numbers and have compare them against BEM numerical simulations, finding excellent agreement. The conditions determining the generation of liquid threads with diameters below 1\(\mu\)m diameter of the injection tube have been analyzed in detail and the crucial role of the inner to outer viscosity ratio for the formation of such tiny jets has been elucidated. Thanks to our numerical results, we deduce a simple model that predicts, as a function of the control parameters, the conditions under which this type of liquid jets are generated as well as the diameters of the resulting drops.

2:47PM QQ.00010 Novel high bandwidth wall shear stress sensor for ultrasonic cleaning applications, S. ROBERTO GONZALEZ-AVILA, FIRDHAUS PRABOWO, Nanyang Technological University, CLAUS-DIETER OHL, Nanyang Technological University — Ultrasonic cleaning is due to the action of cavitation bubbles. The details of the cleaning mechanisms are not revealed or confirmed experimentally, yet several studies suggest that the wall shear stresses generated are very high, i.e. of the order of several thousand Pascal. Ultrasonic cleaning applications span a wide range from semiconductor manufacturing, to low pressure membrane cleaning, and in the medical field cleaning of surgical instruments. We have developed a novel sensor to monitor and quantify cleaning activity which is (1) very sturdy, (2) has a high bandwidth of several megahertz, (3) is cheap in manufacturing costs, and (4) of very small size. We analyze the sensor signal by comparing its response time correlated to single laser induced cavitation bubbles using high-speed photography. Additionally, we will present first measurements in ultrasonic cleaning baths using again high-speed photography. A preliminary discussion on the working mechanism of the sensor will be presented.

\(^1\)EWI and MOE Singapore
12:50PM QR.00001 Fluid-thermal validation of a high-fidelity multi-physics computational tool1. CHRISTOPHER OSTOICH, DANIEL BODONY, PHILIPPE GEBBELLE, University of Illinois at Urbana-Champaign — In order to efficiently design any vehicle, a detailed knowledge of the potential conditions it will see during operation is essential, especially for sustained hypersonic flight. Traditionally, experimental research supplemented with semi-analytical modeling provided the necessary, reliable information to create effective designs. Recently, designs are being pushed outside the existing experimental datasets available for model calibration and there exists debate whether new experiments are possible. There is thus more emphasis being placed on computation-based investigations of the performance of various subsystems. Without sufficient experimental evidence to validate the computational model, notable challenges are encountered. We present results from a combination of experimental and computational analysis of a hypersonic aero-engine test case. For an adiabatic wall, the near-wall structure exhibits a variable-property effect. Using the mean density to scale disturbances associated with individual pores.

1Supported by the U.S. Air Force Research Laboratory Air Air Directorate under contract number FA8650-06-2-3620.

1:03PM QR.00002 Porous coatings for hypersonic laminar flow control, MATTHEW INKMAN, GUILLAUME BRES, TIM COLONIUS, California Institute of Technology, ALEXANDER FEDOROV, Moscow Institute of Physics and Technology — We present the results of linear and nonlinear simulations of hypersonic boundary layers over ultrasonic absorptive coatings consisting of uniform arrays of rectangular pores. Through direct numerical simulation of the two-dimensional Navier–Stokes equations, we explore the effects of coatings of various porosities and pore aspect ratios on the growth rate of the second mode instability. The performance of deep pores operating in the attenuative regime, in which acoustic waves are attenuated by viscous effects within the pores, is contrasted with more shallow pores operating in the cancellation/reinforcement regime. The results of linear simulations in many cases match the results of linear stability theory and confirm the ability of such coatings to stabilize the second mode. At certain conditions such as high porosity and large acoustic Reynolds numbers, the porous layer leads to instability of slow waves, introducing a new instability due to coupled resonant forcing of the cavity array. We confirm the observed instability arises in the linear stability theory, and suggest constraints on cavity size and spacing. Finally, nonlinear simulations of the same geometries confirm the results of our linear analysis; in particular, we did not observe and “tripping” of the boundary layer due to small scale disturbances associated with individual pores.

1:16PM QR.00003 A numerical study of compressible turbulent boundary layers, MAHER LAGHA, JOHN KIM, JEFF ELDREDGE, XIAOLIN ZHONG, Ucla — Compressible turbulent boundary layers with free-stream Mach number ranging from 2.5 up to 20 are analyzed by means of direct numerical simulation of the Navier–Stokes equations. The simulation generates its inflow condition using the rescaling-recycling method. The main objective is to study the effect of Mach number on turbulence statistics and near-wall turbulence structures. The present study shows that the main turbulence statistics can be correctly described as variable-density extensions of incompressible results. We show that the apparent increase in the magnitude of the fluctuating Mach number with increasing free-stream Mach number is a variable-property effect. Using the mean density to scale the fluctuating Mach number collapses results for different freestream Mach number. The increase in the pdf tails of the dilatation is also shown to be a variable-property effect. Compressible boundary layers are shown to be similar to incompressible boundary layers in that, without the linear coupling term, the turbulence cannot be sustained. The linear coupling term is necessary to generate the wall-layer streaks. For an adiabatic wall, the near-wall structure exhibits the same characteristics as in incompressible turbulent flow in terms of the spanwise spacing of the streaks (≈ 100+). For isothermal walls, near-wall turbulence structures show their dependence on the surface heat flux.

1:29PM QR.00004 Estimation of Measurement Errors in Supersonic Wall-Bounded Flows using CFD-Based Simulated PIV, ROSS BURNS, NOEL CLEMENS, HEESEOK KOO, VENKAT RAMAN, The University of Texas at Austin — Particle-lag and resolution effects of PIV are being investigated by applying PIV processing techniques to synthetic CFD-based particle fields. Used previously by several investigators, this method aids in identifying sources of error in the flow and in the comparison of experimental and computational data. The technique utilizes time-resolved DNS or LES data and a modified Stokes flow model of particle motion to create a particle field and generate pairs of synthetic PIV images. A simulated Mach 5 inlet-isolator flow exhibiting multiple shock reflections and significant separation is used as a test application; simulated TiO2 particles ranging in size from 0.1 μm to 2.0 μm are distributed uniformly within the model geometry and evolved in time. In agreement with previous experimental results, particle fields rapidly become inhomogeneous in high-gradient regions, with increased inhomogeneity occurring with greater particle size. A preliminary comparison of synthetic PIV-derived velocity fields with those from the LES data indicate significant errors in regions of high gradients, in particular shocks and regions of separated flow, with higher particle inertia yielding increasing errors.

1:42PM QR.00005 Transition of high-speed flow induced by roughness elements, PRAHLADH IYER, SUMAN MUPPIDI, KRISHNAN MAHESH, University of Minnesota — Transition induced by isolated and distributed roughness elements at supersonic speeds is studied using DNS on unstructured grids. Flow past a hemispherical bump placed on a flat plate is simulated for three Mach numbers [3.37, 5.26, 8.23] with simulation parameters chosen to match the experiments carried out by Daney et al. (AIAA-2009-394). Unsteady flow structures were observed for Ma=3.37, 5.26 while Ma=8.23 remained laminar downstream of the trip. Qualitative comparison between the computation and experiment show good agreement. Based on the computed skin friction coefficient values, Ma=3.37 appeared to become turbulent in nature, Ma=5.26 was transitional and Ma=8.23 was laminar. The effect of distributed roughness on transition was studied at Ma=2.9. A laminar boundary layer at Ma=2.9 was observed to transition to a turbulent boundary layer that shows good quantitative agreement with experimental data. The free-stream Mach number and roughness amplitude were seen to strongly influence whether or not the flow transitions. A local Reynolds number based on bump/roughness amplitude is seen to correlate the tendency to transition for both single bump and distributed roughness cases.

1:55PM QR.00006 Transition delay in hypervelocity boundary layers via CO2 injection, J.S. JEWELL, California Institute of Technology, I.A. LEVYA, Air Force Research Laboratory, N.J. PARZIALE, H.G. HORNUNG, J.E. SHEPHERD, California Institute of Technology — A novel method to delay transition in hypervelocity flows in air over slender bodies by injecting CO2 into the boundary layer is demonstrated and investigated. Experimental data were obtained in Caltech’s T5 reflected shock tunnel. The experimental model was a 5 degree half-angle sharp cone instrumented with thermocouples, providing heat transfer measurements from which transition locations were determined by comparison with laminar and turbulent heat flux correlations. An appropriate injector was designed, and the efficacy of injecting CO2 in delaying transition was gauged at various mass flow rates, and compared with both no injection and Ar injection cases. At an enthalpy of approximately 6 MJ/kg, no transition delay due to CO2 was observed, but for an enthalpy of approximately 10 MJ/kg, transition delays of up to 100% in terms of Reynolds number were repeatedly documented.
2:08PM QR.00007 Laser-Induced Fluorescence Imaging of Droplet Vaporization and Fuel Dispersion in Supersonic Flow1. Y.J. KIM, R.G. CERFF, J.C. HERMANSOHN, University of Washington — The disruption of simulated fuel droplets in supersonic flow is examined experimentally in a draw-down wind tunnel. Monodisperse 100 μm diameter fluid droplets of 2-propanol and a 50/50 by volume hexanol/pentane (Hex-Pen) mixture are generated upstream of the tunnel entrance. The Hex-Pen droplets potentially become superheated as the local static pressure drops below the vapor pressure. The droplets achieve supersonic velocities relative to the surrounding air, a relative Mach number as high as 1.8 and Weber numbers as high as 300. Laser-Induced Fluorescence imaging of the disrupting droplets and the expelled vapor was performed with a pulsed 266 nm Nd:YAG laser. Droplets containing 5% acetone were illuminated by a laser light volume sufficient to capture the entire disrupting droplet. The dispersion of the expelled vapor indicates that Hex-Pen droplets evaporate more rapidly with downstream distance than the non-volatile 2-propanol droplets. The degree to which the vaporization rate for the Hex-Pen droplets exceeds that of the 2-propanol droplets increases at the point downstream of throat where superheating appears to commence.

3Supported by National Science Foundation

2:21PM QR.00008 Experimental investigation of compressibility effects in a separated boundary layer in supersonic flow. JEAN-PAUL DUSSAUGE, IUSTI / Univ. Aix-Marseille / CNRS UMR 6595, SÉBASTIEN PIPONNIAT, INSTITUT Pprime, PIERRE DUPONT, IUSTI / Univ. Aix-Marseille / CNRS UMR 6595 — The structure of the mixing layer formed at the edge of a separation bubble, in a supersonic boundary layer subjected to an impinging oblique shock wave is explored experimentally. An estimation of the spreading rate, based on PIV measurements of velocity variance is proposed. It is shown that, in spite of the rather high convective Mach number produced in the separated zone, the rate of spread of the mixing layer is quite large. It is checked this is only the result of the orientation of the layer with respect to the surrounding flow: this increases significantly the mass entrainment. Moreover, it is shown that the resulting level of turbulent friction is just adapted to this spatial growth rate. It is concluded that, as a first approximation, the behavior of this shear layer follows the physics of the canonical compressible mixing layer.

2:34PM QR.00009 A study of entropy rise across supersonic pressure exchange enhancing rotors, KARTIK BULUSU, CHARLES GARRIS, The George Washington University — Pressure exchange can be envisioned as a process where work is done by a fluid with high kinetic energy on another fluid with relatively low kinetic energy by utilizing the non-steady pressure forces at the fluid-fluid interface in the laboratory frame of reference. A novel supersonic ejector based on this process was conceptualized and offers non-dissipative flow induction, improved efficiency and environmental benefits. Entropy production in such devices holds the key to any efficiency improvements and therefore, entropy generation from flow structures such as oblique shocks under supersonic flow conditions was studied using schlieren photography. Oblique shocks emanating from the apexes of three cone-vane type of rotors (Truncated Ramp Vane, Ramp Vane and Double cone type) of different semi-cone vertex angles (20 deg., 10 deg., 25 deg. respectively), designed to produce pressure exchange were captured for upstream mach numbers (M=1.5, 1.75, 2) in air. Entropy rise across the oblique shocks was estimated from shock angle measurements and compared to a theoretical entropy rise. Analysis revealed that Double Cone Rotor produced three orders of magnitude higher entropy rise than the Ramp Vane Rotor. Furthermore, an increase in entropy rise (approximately 0.5 orders of magnitude) due a small angle of attack (2.5 deg.) was observed in the Ramp Vane Rotor.

2:47PM QR.00010 High-Energy Molecular Beam Source Using a Non-Diaphragm Type Small Shock Tube. YUTA YOSHIMOTO, NOBUYA MIYOSHI, IKUYA KINEFUCHI, KAZUYA SHIMIZU, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — The molecular beam technique is one of the powerful tools to analyze gas-surface interactions. In order to generate high-energy molecular beam in a range of 1 - 5 eV, which corresponds to the typical activation energy of surface reactions, we are developing a beam source using a non-diaphragm type shock tube, which can operate at a repetition rate high enough for efficient data acquisition. We made the volume of a tube much smaller than that of conventional ones so that the evacuation time between each shot becomes as short as possible. Our measurement of shock Mach numbers showed that even small diameter (2 or 4 mm) tubes, in which the wall boundary layer has a large influence on the propagation of shock waves, could generate molecular beam with the translational energy of more than 1 eV. This is because the reduction of shock formation distance by rapid opening of the valve, which separates a high pressure room from a low pressure room, weakened the effect of viscous damping on the accelerating shock wave. In addition, the convergent shock tubes of which diameters linearly decrease from 4 to 2 mm exhibited higher Mach numbers than straight ones. This indicates that the application of the convergent tube with the optimized geometry would be promising for generating high-energy molecular beam.

Tuesday, November 23, 2010 12:50PM - 3:00PM –
Session QS.0001 X: Micro-Swimming Collective Behavior
Grand Ballroom A

12:50PM QS.00001 Collisional low velocity phase of concentrated rod-shaped bacteria1. LUIS CISNEROS, JOHN KESSLER, University of Arizona, SUJOY GANGULY, RAYMOND GOLDSTEIN, University of Cambridge — Suspensions of self propelled wild-type Bacillus subtilis exhibit a transition from independent motion at low concentration to a complex collective dynamics at large concentrations, as a consequence of steric and hydrodynamic interactions. The collective phase displays domains with velocities higher than those of individual swimming cells, correlated with strong co-directionality, termed Zooming BioNematic (ZBN) phase. At intermediate concentrations we find a regime where intercellular collisions, characterized by stopping followed by reconstitution of the propulsion mechanism, produce reduction of mean swimming speeds considerably below that observed for free individual cells. This transitional phase is termed “the jamming phase” by analogy with concentrated automobile or pedestrian traffic. In this regime cell-cell separations are sufficiently small to produce a high frequency of collisions, but not small enough to trigger collective organization. A basic model that considers the typical acceleration of bacteria after collisions and the associated mean free time as a function of cell concentration is shown to yield the observed reduction of swimming speed in the jamming phase.

1DOE W31-101-ENG38

1:03PM QS.00002 Chemotaxis affects hydrodynamics in suspensions of micro-swimmers. ENKELEIDA LUSHI, MICHAEL SHELLEY, Courant Institute, New York University — Microorganisms are known to respond to a dissolved chemical substance by moving preferentially away or toward its source. We study such chemotactic responses at the population level when micro-swimmers are hydrodynamically coupled. To do this we couple a recently developed kinetic model of motile suspension dynamics with a field equation for a chemical substance that diffuses and is advected by the large-scale fluid flows created by the micro-swimmers. We also allow this substance to be produced or consumed by the swimmers themselves. Two models of chemotactic response are considered. One is a simple model for an organism smoothly turning, while moving at constant speed, to align with a chemical gradient. The second is a previously developed model of the effect of modulated run-and-tumble dynamics by individual swimmers. We investigate the linear stability of nearly isotropic suspensions for both models by considering both Pusher micro-swimmers and Pullers. An instability due to chemotaxis is shown to occur in a band of perturbation wavelengths. Nonlinear dynamics are investigated using numerical simulation in two dimensions. We observe aggregation and possible concentration divergences in suspensions of Pullers and the formation of mixing flows in suspensions of Pushers. In the latter case we observe that chemotaxis slows and modifies the mixing dynamics of the system.
Experimental tests predict that “pullers” should instead have a higher viscosity than non-motile suspensions. Additionally, we observe shear-induced migration of active suspensions. Suspensions of “pullers” have a comparatively lower effective viscosity than passive suspensions. This observation contrasts recently proposed theories which characterize the dynamic shear viscosity for both motile and non-motile suspensions of Chlamydomonas reinhardtii. Here we show that the conditions under which the rheology of such suspensions is characterized by two dimensionless parameters, measuring cell stability and swimming speed. We find that turbulent flow separates different species into spatially distinct patches and rationalize these predictions with a simple model of von Karman flow. Preliminary experiments support model results. By reducing the mean distance between organisms, this previously unconsidered mechanism can markedly increase encounter rates, which shape all ecological interactions in the Ocean.

We acknowledge support from the EPSRC, BBSRC, the Marie-Curie Program (M.P.), and the Schlumberger Chair Fund.
the cylinder arrays, having strong implications for fish behavior in such environments.

variety of flow regimes. The results show a strong interaction between the turbulent boundary layer created by the roughness elements and the wakes behind

how a rough turbulent boundary layer interacts with the flow structures created by obstacles (cylinders arrays) in the channel. PIV data were acquired for a

experimental studies to further apply the results from our previous field observations (Cotel et al. 2005) and current laboratory experiments by determining

context of fish responses to turbulent fluctuations. The research performed at IMFT under the umbrella of the NSF IREE grant used complimentary laboratory

control systems and greater ability to generate propulsive power to maneuver. There are direct engineering applications of such work: the design of fishways,

maximum attainable vortex strength.

oscillation frequency, and thus Strouhal number, affects vortex interaction and is found to significantly modify the resulting velocity profiles in the wake of the

observed as the traveling wave wavelength is decreased, which corresponds to a decrease in efficiency as reported by Clark and Smits (2006). Alteration of the

for a wide spectrum of oscillation frequencies and traveling wave wavelengths are identified. A bifurcation from a 2S wake structure to a 2P wake structure is

the vortical patterns and structures developed in the wake of a manta ray-like fin. A DC servo motor powers a gear train to produce the traveling wave motion;

University, ANTOINE CARRIOU, Ecole Superieure de Physique et de Chimie Industrielles, ALEXANDER J. SMITS, Princeton University — Batoid fish such as

supported through MURI ONR N00014-08-1-0642.

1:16PM QT.00003 Vortex Formation in the Starting Flow of Rotating Low-Aspect-Ratio Plates , ADAM DEVORIA, MATTHEW RINGUETTE, State University of New York at Buffalo — We investigate the unsteady flow of fish fin-like plates accelerating from rest through various angular velocity profiles. The objective is to gain an understanding of the connection among the prescribed kinematics and resulting vortex formation; a relationship which has not currently been thoroughly explored. The root-to-tip flow that is induced by the plate motion is expected to have significant effects on the vortex formation. Additionally, different plate shapes are studied to compare the effects of geometrical changes. The experiments are conducted in a water tank, and the plates have a fixed axis of rotation. Digital particle image velocimetry (DPIV) is used to measure the flow velocity in a symmetry plane through the plates. Vorticity and circulation are subsequently computed and vortices are distinguished from surrounding flow structures using vortex identification schemes. Carefully incorporating these techniques will aid the development of scaling laws to characterize the vortex formation with maximum attainable vortex strength.

1:29PM QT.00004 Vortical structures in the wake of an undulating fin , PETER A. DEWEY, Princeton University, ANTOINE CARRIOU, Ecole Superieure de Physique et de Chimie Industrielles, ALEXANDER J. SMITS, Princeton University — Batoid fish such as the manta ray propel themselves through the water by producing a traveling wave motion along the chord of their pectoral fin. Such a motion produces thrust through the development of an unsteady vortex street that results in a jet-like average flow. Digital particle image velocimetry (DPIV) is used to characterize the vortical patterns and structures developed in the wake of a manta ray-like fin. A DC servo motor powers a gear train to produce the traveling wave motion; whose frequency and wave length can be varied. The amplitude of the traveling wave motion linearly increases along the span of the fin. Wake morphologies for a wide spectrum of oscillation frequencies and traveling wave wavelengths are identified. A bifurcation from a 2S wake structure to a 2P wake structure is observed as the traveling wave wavelength is decreased, which corresponds to a decrease in efficiency as reported by Clark and Smits (2006). Alteration of the oscillation frequency, and thus Strouhal number, affects vortex interaction and is found to significantly modify the resulting velocity profiles in the wake of the fin. Notably, increasing the Strouhal number beyond optimal conditions, reported by Clark and Smits, corresponds to a reduction in the extent that the jet-like average flow is observed downstream of the fin.

1:42PM QT.00005 Characterization of the interaction between a rough boundary layer and multiple cylinders wakes , ALINE COTEL, University of Michigan, OLIVIER EIFF, IMFT, PRATIK PRADHAN, University of Michigan — Among many ecologically important aspects of fish locomotion, turbulence is thought to create large stability challenges for fishes. Turbulence is a ubiquitous, highly variable feature of aquatic habitats (Denny 1988). Species that are more prevalent in “energetic water” (high flow, high turbulence) have more effective control systems and greater ability to generate propulsive power to maneuver. There are direct engineering applications of such work: the design of fishways, fish ladders, culverts, etc. No work to date has explored the interaction of a rough boundary layer (typical of natural environments), with wake flows in the context of fish responses to turbulent fluctuations. The research performed at IMFT under the umbrella of the NSF IREE grant used complimentary laboratory experimental studies to further apply the results from our previous field observations (Cotel et al. 2005) and current laboratory experiments by determining how a rough turbulent boundary layer interacts with the flow structures created by obstacles (cylinders arrays) in the channel. PIV data were acquired for a variety of flow regimes. The results show a strong interaction between the turbulent boundary layer created by the roughness elements and the wakes behind the cylinder arrays, having strong implications for fish behavior in such environments.
software. The temporal evolution of the invariants of motion (circulation, energy and impulse) was determined from the DPIV velocity field. The invariance of the vortex ring separation. We fabricated a prototype vortex ring generator which controls the jet diameter and jet velocity independently. This device was used to observe in the early stages of the jump maneuver and was often on the order of 5 to 15 times gravity. Correlations between the maximum energy, power in, number of tail beats, jump height and overall jumping kinematics will be discussed.

2:08PM QT.00007 The effect of a variable diameter nozzle on starting jet formation and separation dynamics, MIKE KRIEG, CU, KAMRAN MOHSENI — As a jet is forced through a nozzle, the shear layer formed at the interface rolls back on itself forming a vortex ring. At a critical point the circulation of the leading vortex ring becomes saturated causing it to “pinch-off” from the trailing shear flow, which then forms a wake of trailing vortices. Jet separation occurs at a nearly universal formation time (Gharib et. al. 1998). Both squid and jellyfish utilize the large impulse transfer associated with vortex ring formation to propel. Both swimmers are also known to actively change the diameter of the fluid interface during jetting. It was predicted by Mohseni et. al. (2001) that changing the diameter of the shear layer during formation can delay the vortex ring separation. We fabricated a prototype vortex ring generator which controls the jet diameter and jet velocity independently. This device was configured to fire a jet through a nozzle which was both opened and closed at a constant rate. The fluid driving mechanism was configured to compensate for the nozzle deformation and maintain a constant jet velocity. The jet formation dynamics were captured using a high speed camera and commercial DPIV software. The temporal evolution of the invariants of motion (circulation, energy and impulse) was determined from the DPIV velocity field. The invariance principle demonstrated a high accuracy before ring saturation, with some losses due to viscosity.

2:21PM QT.00008 Caltech’s Fish-inspired Wind Farm: Results from the first summer1, JOHN DABIRI, California Institute of Technology — Field tests are being conducted at the Caltech Field Laboratory for Optimized Wind Energy to study the aerodynamic interactions of vertical-axis wind turbines in closely-spaced arrays. A model of the wind farm performance—inspired by previous mathematical models of fish schooling—suggests that substantially higher power per unit land area can be extracted relative to existing wind farms of horizontal-axis wind turbines by tuning the spatial arrangement of the turbines. Results from the first summer of field testing support the conclusions of the model, while indicating opportunities for further refinements of the model.

1Funding from the National Science Foundation and US Department of Energy is gratefully acknowledged.

2:34PM QT.00009 Effect of Noise and Flow Field Resolution on the Evaluation of Fluid Dynamic Forces on Bodies Using only the Velocity Field and its Derivatives1, MARIA CECILIA BREA, PAUL S. KRUEGER, Southern Methodist University — Determining unsteady fluid dynamic forces on bodies using only measurements of the velocity field and its derivatives is essential in many investigations, including studies of freely swimming or flying animals. In this project, all terms in a control-volume force equation utilizing only the velocity field and its derivatives discussed by Noca et al. (J. Fluids Struct., 13, 551 - 578) will be analyzed with regard to the influence of flow field noise and resolution to determine which terms dominate the error in the computed force and which factor has the greatest effect on the error. Using analytical and computational flow fields for which the lift and drag forces are known, irregularities found in real experimental results including noise and reduced spatial/temporal resolution will be added to assess their effect on the computed forces. Results for several canonical flows will be presented.

1Support by the SMU Department of Mechanical Engineering is greatly appreciated.
We present experimental studies of the propagation of a reaction front in a fluid flow composed of a chain of alternating vortices. We propose that the tools used to describe the transport of a passive impurity in a flow can be expanded to account for the behavior of a reaction front. In particular, we propose that motion of a reaction front from one region to another in the flow is determined by burning manifolds and burning lobes. These ideas are tested experimentally for both the time-independent and time-dependent vortex chain. For a time-independent flow, the time that it takes for a triggered reaction to propagate from one vortex to the next is the minimum time $\tau$ for the stable burning manifold $BS(\tau)$ to envelope the original trigger point. For a time-dependent (oscillatory) vortex chain, we use the burning manifold/lobe framework to explain mode-locking behavior seen in earlier studies.

1Supported by NSF Grants DMR-0703635, DMR-1004744 and PHY-0552790.

2Current address: Truman State University, Kirksville, MO.

**1:55PM QU.00006 Numerical Investigation of the Hydrogen Jet Flammable Envelope Extent with Account for Unsteady Phenomena**

BORIS CHERNYAVSKY, PIERRE BENARD, Institut de recherche sur l’hydrogene, UQTR — An important aspect of safety analysis in hydrogen applications is determination of the extent of flammable gas envelope in case of hydrogen jet release. Experimental investigations had shown significant disagreements between the extent of average flammable envelope predicted by steady-state numerical methods, and the region observed to support ignition, with proposed cause being non-steady jet phenomena resulting in significant variations of instantaneous gas concentration and velocity fields in the jet. In order to investigate the influence of these transient phenomena, a numerical investigation of hydrogen jet at low Mach number had been performed using unsteady Large Eddy Simulation. Instantaneous hydrogen concentration and velocity fields were monitored to determine instantaneous flammable envelope. The evolution of the instantaneous fields, including the development of the turbulence structures carrying hydrogen, their extent and frequency, and their relation with averaged fields had been characterized. Simulation had shown significant variability of the flammable envelope, with jet flapping causing shedding of large scale rich and lean gas pockets from the main jet core, which persist for significant times and substantially alter the extent of flammability envelope.

**2:08PM QU.00007 Water Mist Interaction with Flame Spreading Against Gravity**

CHENTHIL KUMAR, AMIT KUMAR, IIT Madras — Water mist fire-suppression systems have gained importance since chemical agents like Halons are being phased out for environment preservation. The present study focuses on the effect of water mist droplets size and concentration in inhibiting the flame spreading downward over thin solid fuel at different gravity levels. The water droplets are introduced into the air stream at pre-specified concentration and droplet size. An Eulerian-Eulerian two phase model is used for this particular study. The polydisperse spray is modeled using the moments of the droplet size distribution function. The gas phase is modeled by full Navier-Stokes equations for laminar flow along with the conservation equations of mass, energy & species. A one-step Arrhenius reaction between fuel vapor and oxygen is assumed. The gas radiation equation is solved using DOM. The solid fuel considered is assumed to burn ideally to form fuel vapors without melting. The thin solid fuel is modeled by equations of continuity and energy. The pyrolysis of fuel is modeled as one-step, zeroth-order Arrhenius kinetics. For the dilute sprays, droplet sizes below 100$\mu$m are increasingly effective in reducing the flame temperature.

**2:21PM QU.00008 Vorticity, Strain Rate, and Scalar Gradient Dynamics in Premixed Reacting Flows**

PETER HAMLINGTON, ALEXEI POLUDENKO, ELAINE ORAN, Naval Research Laboratory — The interactions between turbulence and flames in premixed, stoichiometric hydrogen-air combustion are studied as a function of turbulence intensity by analyzing the coupled dynamics of the vorticity, strain rate, and scalar (reactant mass fraction) gradient. The analysis is based on fully compressible numerical simulations of statistically planar flames at a range of intensities, where the intensity is characterized by the turbulent rms velocity in the unburned mixture with respect to the laminar flame speed. The simulations have been carried out using the reactive-flow code Athena-REX, and high numerical resolution allows the dynamics within the flame to be studied using conditional diagnostics based on the local, instantaneous value of the scalar. Particular emphasis is placed on the magnitudes and relative alignments of the vorticity, strain rate, and scalar gradient, which give insights into the interactions between these quantities in the presence of heat release effects. The analysis shows that there are substantial variations in the dynamics with both the turbulence intensity and position in the flame. The implications of these results for understanding the structure and evolution of premixed flames, particularly when the turbulence intensity is large, are discussed.

1This work was supported, in part, by the NRC Research Associate Program, AFOSR, NRL, ONR, and by NSF through the TeraGrid resources.

**2:34PM QU.00009 Characteristics of Edge Flames in Microcombustors**

JOANNA BIERI, University of Redlands, MOSHE MATALON, University of Illinois at Urbana-Champaign — Two streams, one containing fuel and the other oxidizer, are flowing into a relatively narrow channel where they mix and support an edge flame at some distance downstream. Our analysis is based on two models; one that fully couples the fluid dynamics and transport equations, used to determine the flame shape and location, and the other that assumes a constant-density flow, used to test the steady solutions for stability. It is found that in relatively wide channels the flame has a premixed, rounded edge with a trailing diffusion flame, but when the channel width decreases the flame is located further away from the supply and has a broader edge that can span the entire channel, when its width becomes comparable to the characteristic flame thickness. The effect of thermal expansion is to relocate the edge flame closer to the reactant supply. Heat losses at the channel walls cause a drop in the overall temperature and, as a result, the edge flame is confined to the center of the channel and the trailing diffusion flame is shortened significantly. Depending on the Lewis number, the flow rate, and the extent of heat loss, the edge may either remain steady, oscillate, or be blown off by the flow. With appreciable heat losses, residual fuel and oxidizer are observed at the end of the channel, so that under appropriate conditions, they could re-ignite and support a streak of diffusion flamelets, as seen experimentally.

**Tuesday, November 23, 2010 12:50PM - 3:00PM**

**Session QV Suspensions II**

**Hyatt Regency Long Beach Regency B**

**12:50PM QV.00001 Mixing and microstructure in sedimenting suspensions**

ELISABETH GUAZZELLI, LAURENCE BERGOUGNOUX, IUSTI CNRS Polytech Marseille — We present experiments concerning the statistics of particle microstructure in sedimenting suspensions. We explore the role of initial mixing on the microstructure and its impact on velocity fluctuations.
1:03PM QV.00002 Micro rheology of colloidal dispersions, ROSEANNA ZIA, JOHN BRADY, California Institute of Technology — Dilute and concentrated colloidal dispersions are studied via nonlinear micro rheology in the presence of excluded volume and hydrodynamic interactions via Stokesian dynamics simulation. In nonlinear micro rheology, the motion of a Brownian probe is tracked as it is driven by an external force through the suspension. Mean probe motion defines the micro viscosity by application of Stokes’ drag law; probe fluctuations due to collisions give rise to force-induced diffusion. Together, these two quantities define the particle stress, from which the normal stress differences and osmotic pressure are obtained. Rheological properties depend strongly on the deformed microstructure, which in turn depends on the strength with which it is driven from equilibrium by the probe, as given by the Pecelet number - the strength of probe forcing compared to thermal forces: $P_e=F_b/kT$, where $kT$ is the thermal energy and $b$ the bath particle size. Hydrodynamic interactions strongly influence this structure, modulating the force-induced diffusion and giving rise to force-thickening at large Pe, thus altering the effects of viscous stress at $Pe>>1$.

1:16PM QV.00003 Plugging of microchannels by spherical particles, ERIC CLIMENT, Institute of Mecanique des Fluides de Toulouse, CONSTANT AGBANGLA, PATRICE BACCHIN, Laboratoire de Genie Chimique, UNIVERSITY OF TOULOUSE COLLABORATION, FEDERATION DE RECHERCHE CNRS - FERMAT COLLABORATION — We investigate by means of numerical simulations the dynamic formation of 3D structures of microparticle aggregates blocking the flow through microchannels. Both the geometries of a straight channel and a sudden reduction of section are analyzed. We use the Force Coupling Method (Climent & Maxey, 2010) to handle simultaneously multi-body hydrodynamic interactions of a confined flowing suspension together with particle/particle and particle/wall surface interactions leading to the adhesion and aggregation of particles. The basic idea of the Force Coupling Method relies on multipole expansion of velocity perturbations induced by the presence of particles in the flow. Simulation results show that varying the magnitude of DLVO interparticle and particle/wall interactions leads to distinct scenarios of pores clogging. We investigate the kinetics of the microchannel occlusion (corresponding to a temporal decrease of the bulk permeability of the channel). We identify the nature of the fouling mechanism: deposition, interception, bridging ... (see the papers of Sharp & Adrian (2005), Ramachandran & Fogler (1999) and Marshal, (2007)). We acknowledge the support of the research federation FERMAT.

1:29PM QV.00004 A Quantitative Study of Bulk Stresses in Nonlinear Micro rheology, RYAN DEPUIT, TODD SQUIRES, University of California, Santa Barbara — We investigate the nonlinear micro rheology of a simple model system - a spherical probe translating through a dilute suspension of rigid rods - to elucidate a variety of issues inherent in the interpretation of nonlinear micro rheology. We have developed a computational system to quantitatively examine the issues present in interpretation of nonlinear micro rheology, as originally discussed by Squires (Langmuir, 2008). Following recent work emphasizing the importance of the microstructural behavior in the bulk (Striam et. al, 2009), we focus our attention on the bulk microstructural deformation, and examine the significance of its (Lagrangian) transient nature, as well as the consequences of the mixed and inhomogeneous flows inherent to nonlinear micro rheology. From this quantitative study, we pose solutions for the current theoretical issues facing nonlinear micro rheology in interpretation and comparison of the micro viscosity with the shear viscosity from traditional bulk rheometry.

1:42PM QV.00005 Long-time self-diffusivity of a catalytic particle in a dilute suspension, SERGEY SHKLYAEV, California Institute of Technology, JOHN F. BRADY, California Institute of Technology, UBALDO M. CORDOVA-FIGUEROA, University of Puerto Rico — Active micro rheology, which studies local changes in a microstructure of a suspension near a forced particle and a feedback of this redistribution on the particle motion, is of keen interest. Implementation of this concept to the chemically active particles is a promising field of research. We consider a long-time self-diffusivity of a catalytic (probe) particle dragged by an external force through a dilute suspension comprising reactant and product particles. The former decay at the contact with the probe particle producing $+$ product particles. Neglecting by the hydrodynamic interaction, we derive the boundary value problem which governs the microstructures of the both types of suspended particles. Distortion of the microstructure due to both the motion of the probe and the chemical reaction leads to changes in the tensor of long-time self-diffusivity. The problem is considered analytically in a several limiting cases and numerically otherwise. In the absence of advection contributions ensuing from reactant and product particles are completely different. The first one is negative and tends to zero for the fast reaction, whereas the second one is not of fixed sign and remains finite in the mentioned limiting case. Advection amplifies both the contributions, the increase is more pronounced for the longitudinal component of the diffusivity tensor.

1:55PM QV.00006 In Situ Observations of Electric Field Induced Nanoparticle Aggregation, T.J. VOEHL, N.D. BROWNING, W.D. RISTENPART, Dept. Chem. Engr. Mat. Sci., Univ. California at Davis — Nanoparticles have been widely observed to aggregate laterally on electrodes in response to applied electric fields. The mechanism driving this behavior, however, is unclear. Several groups have interpreted the aggregation in terms of electrohydrodynamic or electroosmotic fluid motion, but little corroborating evidence has been presented. Notably, work to date has relied on post situ observations using electron microscopy. Here we present a fluorescence microscopy technique to track the dynamics of nanoparticle aggregation in real-time. Using this approach we observe the reaction of 20-nm polystyrene nanoparticles are observed to form optically visible aggregates in response to an applied AC field. Although single particle resolution is lost, the existence of aggregates on the electrode surface is marked by growing clusters of increasingly bright intensity. We present a systematic investigation of the effects of applied potential and frequency on the aggregation rate, and we interpret the behavior in terms of a mechanism based on electrically induced convective flow.

2:08PM QV.00007 Numerical Simulations of Electrostatically Induced Aggregation and Coalescence in Polydisperse Emulsions, G.R. MAGILL, W.D. RISTENPART, Dept. Chem. Engr. Mat. Sci., Univ. California at Davis — Although electrostatic coalescers have long been used to destabilize emulsions of polarizable droplets, the dynamics of droplet aggregation and coalescence remain poorly understood. The aggregation is believed to be primarily driven by dipolar interactions between droplets, suggesting that increasing the electric field strength should increase the rates of aggregation and coalescence. However, recent evidence suggests that coalescence is inhibited above a critical field strength. Here we numerically investigate the dynamics of aggregation and coalescence of polydisperse emulsions. The simulations are based on the point dipole approximation coupled with pseudo hard sphere repulsion at small separations. Two limiting cases are examined in detail: immediate coalescence upon contact, and perfect stability against coalescence. We compare the numerical results to previous experimental work, and we discuss the implications for optimizations of electrostatic coalescers.

2:21PM QV.00008 Magnetically Guided Propulsion of Osmotic Motors, GLENN VIDAL, CARLOS RINALDI, UBALDO CORDOVA-FIGUEROA — Propulsion of artificial nano- and micro-scale objects induced by chemical reactions is one of the most exciting challenges in colloidal physics. Recent experiments have shown that directed motion of catalytic motors is hindered by their rotary Brownian motion, preventing its potential to be fully realized. The present work investigates the magnetically guided propulsion of a colloidal particle—the osmotic motor—immersed in a dispersion of colloidal ‘bath’ particles subject to an unidirectional magnetic field using Brownian dynamics simulation. The osmotic motor is propelled by a chemical reaction that consumes bath particles over a portion of its surface leading to the formation of aggregates. The non-equilibrium microstructure of bath particles is imprinted on the surface causing it to move to regions of lower bath particle concentration. The strength of the magnetic field is controlled by the Langevin parameter, which physically measures the relative importance of magnetic to Brownian torques, and dictates the directionality of the osmotic motor. The translational self-diffusivity is measured for different reaction speeds, particle sizes, bath particle concentrations, and magnetic dipole orientations. Finally, a theory to determine the long-time self-diffusivity and time-averaged particle velocity is developed and compared to the simulation results.
2:34PM QV.00009 The Rheology and Microstructure of Dense Suspensions of Elastic Capsules1, JONATHAN CLAUSEN, Sandia National Labs, DANIEL REASON, CYRUS AIDUN, Georgia Institute of Technology — We use a recently developed hybrid numerical technique [MacMeccan et al. (2009)] that combines a lattice-Boltzmann (LB) fluid solver with a finite element (FE) solid-phase solver to study suspensions of elastic capsules. The LB method recovers the Navier-Stokes hydrodynamics, while the linear FE method models the deformation of fluid-filled elastic capsules for moderate levels of deformation. The simulation results focus on accurately describing the suspension rheology, including the particle pressure, and relating these changes to changes in the microstructure. Simulations are performed with hundreds of particles in unbounded shear allowing an accurate description of the bulk suspension rheology and microstructure. In contrast to rigid spherical particles, elastic capsules are capable of producing normal stresses in the dilute limit. For dense suspensions, the first normal stress difference is of particular interest. The first normal stress difference, which is negative for dense rigid spherical suspensions, undergoes a sign change at moderate levels of deformation of the suspended capsules.

1Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. DoE’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

2:47PM QV.00010 Solid Particle Erosion in Slug Flow1, NETAJI RAVIKIRAN KESANA, JON THRONEBERRY, BRENTON MCLAURY, SIAMACK SHIRAZI, University of Tulsa — Erosion is a common problem faced by oil and gas industries, and the repair of pipelines damaged by erosion is extremely costly. Therefore measuring erosion under different flow conditions and in different flow geometries is important to help better understand the effect of various parameters on erosion and to provide information to develop protective guidelines for the oil and gas producers. Specifically, this work examines solid particle erosion in multiphase slug flow and the transition to annular flow regime in a 3-inch pipe with solid (sand) particles of different average sizes (20, 150 and 300 microns) and with different liquid viscosities (1cP, 10 cP). The metal loss is measured using intrusive Electrical Resistance (ER) probes which work on the principle of resistivity of the sample and reference elements. Erosion is measured at three different locations in the pipe, two in a bend and one in a straight section. Results demonstrate that metal loss increases by increasing the superficial gas velocity, superficial liquid velocity, or particle diameter; however, metal loss decreases by increasing the liquid viscosity.

1Sponsored by the member companies of Erosion/Corrosion Research Center.

Tuesday, November 23, 2010 12:50PM - 3:00PM — Session QW Experimental Techniques III Hyatt Regency Long Beach Regency C

12:50PM QW.00001 Spatial filtering and end-conduction effects in turbulence measurements using hot wires , ANAND ASHOK, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — We use grid generated homogeneous isotropic turbulence as a benchmark flow to test the effects of spatial resolution on turbulence measurements with hot wires. The grid turbulence is generated in a low speed 2 by 3 closed circuit wind tunnel using a 0.25" square mesh grid placed at the test section inlet. Measurements of the turbulence statistics and spectra downstream of the grid were made using hot-wires with a constant diameter but varying lengths. An empirical correlation for the attenuation of the energy due to spatial filtering is constructed as a function of the parameter l/η. In addition, we show that end conduction effects depend not only on the l/η ratio but also on the wire material and the wire Reynolds number.

1:03PM QW.00002 On the optimal number and sensor arrangement of multi-sensor hot-wire probes to measure the velocity vector and the velocity gradient tensor in turbulent boundary layers , P. VUKOSLAVEVIC, Univ. of Montenegro, J. WALLACE, N. BARATLIS, E. BALARAS, Univ. of Maryland — Although a 3-sensor array should be sufficient to simultaneously measure the three velocity components, and three such arrays can be combined to measure the six velocity gradients in the cross-stream plane, a fourth sensor has been used for probes developed over the last two decades in order to increase measurement accuracy and the uniqueness range of multi-sensor hot-wire probes. A highly resolved turbulent channel flow DNS with Reₜ = 200 was used to investigate the optimal sensor number and arrangement. The sensors were represented as points on the simulation grid, the effective velocity cooling each sensor was determined assuming an ideal sensor response, and the sensor equations were then solved in response to the DNS field to obtain velocity vector and velocity gradient tensor components. It will be shown that the fourth sensor of each of the three arrays is unnecessary for turbulent boundary layer flow measurements if the arrays’ three sensors are arranged judiciously. Requiring only nine sensors instead of twelve is a great advantage with respect to probe construction, size and sensor interference. Results from the types of 12-sensor probes previously used will be compared to those from an optimally designed 9-sensor probe.

1:16PM QW.00003 Two-dimensional Laser-Cantilever-Anemometer with re-designed cantilever chip - measurements and characterization. MICHAEL HOELLING, JAROSŁAW PUCZYŁOWSKI, JOACHIM PEINKE, ForWind, Institute of Physics — University of Oldenburg — We present measurements performed with the two-dimensional Laser-Cantilever-Anemometer (2D-LCA) equipped with a re-designed cantilever chip. The 2D-LCA measures flow velocities by detecting the deflection of a tiny cantilever (about 140 x 40 micrometer) brought into the flow by means of the laser pointer principle. Latest investigations show a combination of both twisting and bending behavior of the cantilever for inclined angles of attack. Measurements for different combinations of angles of attack and velocities result in an unambiguous two-dimensional calibration which allows the 2D-LCA for simultaneous measurements of these two quantities, and of the longitudinal and transversal velocity component respectively. The new cantilever chip has a vertical structure added at the tip of the cantilever in order to increase the sensitivity to flows from angles of attack. We present measurements taken with the new cantilever chip in turbulent laboratory flows in comparison to the old cantilever design and to x-wire anemometry data.

1:29PM QW.00004 Infrared quantum dots as liquid temperature tracers for imaging through silicon1, MYEONGSUB KIM, MINAMI YODA2, Georgia Institute of Technology — Although a number of optical thermometry techniques estimate fluid temperature fields from changes in the lifetime or intensity of the emissions from fluorescent or phosphorescent species, the majority of these techniques rely on imaging optical signals at visible wavelengths. Silicon (Si), commonly used in microelectronics and microelectromechanical systems (MEMS), is however opaque at these wavelengths, and only becomes partially transparent at near-infrared (IR) wavelengths above ≈1.3 μm. Given the lack of fluorescent species with emissions in the near-IR, colloidal nanocrystals, or “quantum dots” (QD), of lead sulfide overcoated with cadmium sulfide using a new process with a diameter of 5.7 nm were investigated as temperature tracers. The emissions around 1.35 μm for these PbS/CdS QD suspended in toluene at an absorbance of 0.45 were found to decrease by about 0.35 μm per °C increase in the suspension temperature T for T = 20 °C above a standard deviation that gave an uncertainty in T of ≈0.3 °C. The overcoating greatly improves the stability of the QD, and the temperature response of these tracers was consistent for suspended samples stored up to 103 days under nitrogen as well as up to 1 day under air.

1Supported by NSF and ONR
2Presenting author
1:42PM QW.00005 Temperature characterization of CdSe/ZnS quantum-dots applied on anodized-aluminum coating , AKIHISA AIKAWA, Sophia University, HIROTAKA SAKAUE, JAXA — We have developed a quantum-dot (QD) based anodized-aluminum temperature-sensitive paint (AA-TSP) as a global temperature sensor. Compared to a conventional TSP, which uses a polymer as a supporting matrix, the AA-TSP can provide a narrow FWHM that provides a potential to create a multi-color TSP. By using anodized aluminum as a supporting matrix, the resultant AA-TSP extends the temperature detection range, which is limited by a conventional supporting matrix of a polymer. The temperature calibration shows that a resultant AA-TSP can detect the temperature from 100 to 500 K. Six different QDs are chosen for temperature characterizations of the AA-TSP, whose luminescent peaks are at 481, 518, 543, 555, 587, and 615 nm in toluene. The temperature sensitivity of the resultant AA-TSP ranges from -0.6 to -1.5 percent/K. With increase in the luminescent peak, the sensitivity increases. An application of the resultant AA-TSP for a global temperature measurement in a supersonic wind tunnel is included in addition to the temperature characterizations.

1:55PM QW.00006 Fluorescence thermometry measurements of wall surface and bulk fluid temperatures1 , MYEONGSUB KIM, MINAMI YODA, Georgia Institute of Technology — Measuring fluid temperature fields at micron-scale spatial resolution is of interest in applications including microelectronic cooling and microfluidics. Fluorescence thermometry (FT), where temperatures are estimated from variations in the emission intensity of various fluorophores, is commonly used to measure liquid temperatures in a variety of flows. Here dual-tracer FT (DFT) where fluorescein (Fl) and sulforhodamine B were volumetrically illuminated was used to measure temperature fields in the Poiseuille flow of water through a heated 1 mm square channel. The average experimental uncertainties in the DFT results are estimated to be < 0.3 °C at a spatial resolution of 30 μm in the image plane at Re = 3.3 and 8.3. Evansen-wave FT (EFT) where only Fl is illuminated by evanescent waves was also used to measure the liquid-phase temperature field within the first 0.3 μm next to the wall with an average uncertainty of < 0.2 °C at a resolution of 10 μm. Comparison with numerical predictions show that the EFT results are effectively the wall surface temperature. Comparison of the DFT data with numerical predictions suggest that the spatial resolution of these data along the optical axis is at least an order of magnitude greater than the depth of field.

2:08PM QW.00007 Response time characterization of fast responding pressure-sensitive paint , TATSUYA OZAKI, HITOSHI ISHIKAWA, Tokyo University of Science, HIROTAKA SAKAUE, JAXA — Response time characterization of a fast responding pressure-sensitive paint (PSP) is important information in measuring an unsteady flow field. PSP is an optical pressure sensor. The luminescent image from the PSP is related to a pressure map. In the previous works, a time delay from a step change of pressure is generally used to characterize the response time. The thickness of the PSP as well as the PSP binding material greatly influences the response time. Because the temperature influences the diffusion or permeation of a PSP binder, it is also an important parameter to influence the response time. We build a shock tube to create a step change of pressure for response time characterization. This can control the temperature of the PSP. We discuss the PSP response times related to the temperature of the binder as well as the binding materials.

2:21PM QW.00008 Motion-cancelled PSP system for obtaining global unsteady fields of a moving object , KENSEUKE MIYAMOTO, TAKESHI MIYAZAKI, Univ of Electro-Communications, HIROTAKA SAKAUE, JAXA — A motion-cancelled PSP system is introduced for obtaining global unsteady fields on a moving object. This system uses a reference- and signal-image simultaneously acquired by a digital camera. Each image is provided by a two-color pressure-sensitive paint (PSP). The luminescent outputs from the PSP are converted to the pressures or the oxygen concentrations. The existing system uses a color CMOS camera. The green and red images of the camera correspond to the reference and signal images, respectively. Due to the spectral overlay of the images, the pressure sensitivity of the existing system is poor (0.13 percent/kPa). To improve the sensitivity, a spectral separation is necessary. As an improved system, we use a two-CCD chip camera, which can select the pass-band filters in front of the chips. The filters can limit the wavelength range of each luminescent image that prevents the spectral overlay. A comparison with the existing system is provided, and the developed system is applied to an oscillating unsteady motion of a flat plate for demonstration.

2:34PM QW.00009 A Study of Surface-Pressure Estimation from Multiline Molecular Tagging Velocimetry Data1 , AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — This study is motivated by the extraction of surface-pressure information from Molecular Tagging Velocimetry (MTV) data in order to correlate unsteady flow structures with surface forces. The approach we take is to integrate the pressure-gradient acting on the wall, which for a stationary surface can be computed from knowledge of the wall vorticity flux. The latter requires calculation of the second derivative of the velocity at the wall, which is generally difficult to estimate accurately from near-wall velocity data. In this work, we seek to address this issue by capitalizing on the unique ability of MTV to provide very fine resolution of single-velocity-component data near the wall. The accuracy of determining the wall vorticity flux (and surface pressure) from such measurements is examined using theoretical solutions of simple flows and numerical databases. The results provide a guide for the selection of the measurement parameters for accurate implementation of the method, as well as shed light on the practical limits of its applicability.

1Supported by AFSOR grant FA9550-10-1-0342.

2:47PM QW.00010 An Assessment of Oil Film Interferometry to Measure Skin Friction , PETER A. MONKEWITZ, EPFL, Lausanne, Switzerland, ANTONIO SEGALINI, JEAN-DANIEL RÜEDI, Univ. Bologna, Forli, Italy — In recent years, the independent measurement of wall shear stress with oil film interferometry has led to a step increase in the understanding of turbulent boundary layers. However, while many arguments depend critically on a precise knowledge of the skin friction, the systematic errors of the oil film technique are not well known. In particular the basic theory underlying the technique has essentially not evolved since it was first proposed by Tanner & Blows (J. Phys. E: Sci. Instrum., vol. 9, 1976, p. 194). The purpose of this study is to elucidate the dominant systematic error of the classical oil film method. We derive the corrections to the basic Tanner & Blows similarity solution for the film development in zero pressure gradient boundary layers and validate the analysis experimentally. This allows to formulate "best practice guidelines" for the oil film technique that help push uncertainties below 1%.

Tuesday, November 23, 2010 12:50PM - 3:00PM — Session QX Industrial Applications II Hyatt Regency Long Beach Regency D
Atmospheric Boundary Layer.

Streamtube control surfaces are computed in order to determine their overall impact and to better understand the energy exchange between the turbine and the arrays of wind turbines placed in the highly turbulent atmospheric boundary layer, these assumptions do not hold. In the current study, PIV data taken in a inlet and outlet of the streamtube. However, the classical analysis assumes ideal flow, neglecting, in particular effects of turbulence. For applications to large describing the flow upstream and downstream of wind turbines. The analysis is used to relate power output to the differences of kinetic energy fluxes at the MENEVEAU, The Johns Hopkins University — Ever since the streamtube concept was introduced by Betz in 1926, the analysis have been widely used for turbines have been performed. The wind turbine cross section design was based on geometrical optimization study of Rahai and Hefazi for increasing contributions of the lift force to the torque, resulting in significant improvements in the performance of a vertical axis wind turbine. The wind turbine was 30 cm in diameter and 75 cm length, with 45 cm diameter end-plates, placed in the spanwise direction above a 26 degree slanted roof at 20 percent from the roof’s highest elevation and one turbine diameter away from the roof surface. The approaching wind velocity was 30 m/sec and the wind turbine RPM was 233. Results indicate nearly 20 percent improvements in the power output, when compared with the corresponding results for a free standing wind turbine. However, the wind turbine operation imposes oscillatory stress on the roof, which could result in structural vibration and damage and noise generation.

1:03PM QX.00002 Ion Wind Generation and its Application to Cooling Device\(^1\). BUMCHANG KIM, SANGHYUN LEE, Pohang University of Science and Technology (POSTECH), YOU SEOP LEE, Samsung Electronics, KWAN HYOUNG KANG, Pohang University of Science and Technology (POSTECH) — Ion wind generation (IWG) has a long history in the field of electrohydrodynamics (EHD). The application of IWG to cooling devices has drawn much attention, mainly because of its extremely low level of acoustic noise emission, compared to the conventional mechanical fan. In this work, we performed a parametric study for geometrical and electrical configurations, electrode materials, and surrounding media such as air, nitrogen, and argon. Wind velocity and volume flow rate are measured with regard to power efficiency, operational voltages, and polarities such as DC+, DC-, and AC. The effect of electrode materials and the surrounding media on the morphological changes of the electrode surface is discussed. This study envisions that the IWG could be a promising cooling mechanism, although there are several issues such as safety and maintenance that need to be addressed.

\(^{1}\)This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) (Grant No. R0A-2007-20080098-0).

1:16PM QX.00003 Numerical Simulations of a Roof-Top Wind Turbine\(^1\). SHAHAB MOAYEDIAN\(^2\), CEERS/CSULB, HAMID RAHAI\(^3\), MAE Dept./California State University, Long Beach — Unsteady numerical simulations of a high efficiency roof-top wind turbine have been performed. The wind turbine cross section design was based on geometrical optimization study of Rahai and Hefazi for increasing contributions of the lift force to the torque, resulting in significant improvements in the performance of a vertical axis wind turbine. The wind turbine was 30 cm in diameter and 75 cm length, with 45 cm diameter end-plates, placed in the spanwise direction above a 26 degree slanted roof at 20 percent from the roof’s highest elevation and one turbine diameter away from the roof surface. The approaching wind velocity was 30 m/sec and the wind turbine RPM was 233. Results indicate nearly 20 percent improvements in the power output, when compared with the corresponding results for a free standing wind turbine. However, the wind turbine operation imposes oscillatory stress on the roof, which could result in structural vibration and damage and noise generation.

\(^{1}\)The study was performed with a grant from the National Science Foundation.

\(^{2}\)Graduate Assistant

\(^{3}\)Professor and Director

1:29PM QX.00004 Evaluating Contributions of Turbulent kinetic Energy Fluxes To Stream-tube Analysis of Wind Turbines in an Array\(^1\). JOSE LEBRON, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, CHARLES MENEVEAU, The Johns Hopkins University — Ever since the streamtube concept was introduced by Betz in 1926, the analysis have been widely used for describing the flow upstream and downstream of wind turbines. The analysis is used to relate power output to the differences of kinetic energy fluxes at the inlet and outlet of the streamtube. However, the classical analysis assumes ideal flow, neglecting, in particular effects of turbulence. For applications to large arrays of wind turbines placed in the highly turbulent atmospheric boundary layer, these assumptions do not hold. In the current study, PIV data taken in a wind tunnel 3x3 tunnel array experiment (Cal et al. 2010. J. Renewable and Sustainable Energy), is analyzed. The fluxes of turbulent kinetic energy at the streamtube control surfaces are computed in order to determine their overall impact and to better understand the energy exchange between the turbine and the Atmospheric Boundary Layer.

\(^{1}\)Research supported by NSF (Project CBET 0730922).

1:42PM QX.00005 Numerical Modeling of Hydrokinetic Turbines and their Environmental Effects\(^1\). TEYMOUR JAVAHERCHI, ALBERTO ALISEDA, University of Washington — Energy extraction from ocean tides via hydrokinetic turbines has recently attracted scientists and engineers attention as a highly predictable source of renewable energy. However, since the most promising locations in terms of resources and proximity to the end users are in fragile estuarine ecosystems, numerous issues concerning the environmental impact of this technology need to be addressed a priori before large scale deployment. In this work we use numerical simulations to study the possible environmental effects of hydrokinetic turbines through their influence on physical flow variables such as pressure and velocity. The velocity deficit created in the turbulent wake of a turbine affects the settling of suspended sediment in the water column and can lead to deposition into artificial patterns that will alter the benthic ecosystem. On the other side of the spectrum, pressure fluctuation through turbine blades and in blade tip vortices can damage internal organs of marine species as they swim through the device, particularly for small juveniles that behave like Lagrangian trackers. We present sedimentation statistics to understand the sensitivity of this phenomena to turbine operating conditions and sediment properties. We also show pressure history for slightly buoyant Lagrangian particles moving through the turbine and correlations with damage thresholds obtained from laboratory experiments.

\(^{1}\)Supported by DOE through the National Northwest Marine Renewable Energy Center.

1:55PM QX.00006 Hydrodynamic Performance of a Wave Energy Converter\(^1\). YINGCHEN YANG, University of Texas at Brownsville — To harvest energy from ocean waves, a new wave energy converter (WEC) was proposed and tested in a wave tank. The WEC freely floats on the water surface and rides waves. It utilizes its wave-driven angular oscillation to convert the mechanical energy of waves into electricity. To gain the maximum possible angular oscillation of the WEC under specified wave conditions, both floatation of the WEC and wave interaction with the WEC play critical roles in a joint fashion. During the experiments, the submersion condition of the WEC and wave condition were varied. The results were analyzed in terms of the oscillation amplitude, stability, auto-orientation capability, and wave frequency dependency.
2:08PM QX.00007 Energy extraction from flexible slender bodies. KIRAN SINGH, SEBASTIEN MICHELIN, EMMANUEL DE LANGRE, ENSCP Yves, Palaiseau, France — Long slender structures such as underwater cables may develop instabilities that can lead to large amplitude oscillations, chaotic behaviour invariably leading to failure. Whilst the norm is to avoid such regimes, here we examine the converse problem of energy extraction from flexible slender bodies in a fluid. We derive the terms for this fluid-structure interaction problem paying specific attention to large-amplitude deflections in the small curvature limit. We use a local approach to model inviscid and viscous fluid dynamic contributions. We represent the structure as a bi-articulated cylindrical pendulum with stiffness and structural damping introduced discretely at the joints; the simple system thus has two degrees of freedom, (θ₁, θ₂) with fixed-free boundary conditions. We solve the coupled system of second order in time non-linear ODEs and examine the response of this fourth order system, Y = (θ₁, θ₂, θ₁, θ₂)ᵀ. A key objective of this work is to examine the feasibility of energy extraction from such systems, represented by the structural damping term. In order to examine the parameter space for likely solutions, we quantify the associated energies and examine the related problem of stability. In particular, we are interested in seeking stable limit-cycle oscillations. Time permitting, we consider the flexible slender body response for specific cases.

2:21PM QX.00008 On the flow around a hydrofoil close to a permeable wall. MATS NIGAM, Noss — In the pulp and paper-making industry, pressurized screens are used to purify the pulp and to protect the wires on the paper machine. The appertures (slots or holes) on the screen are kept clean from fibers and shives by foils rotating close to the permeable surface. The leading-order theory of a “wing in extreme ground effect” has been modified to take into account the permeability of the screen-plate. Methods for predicting flow separation and for modifying the potential-flow solution due to the same are discussed, and numerical simulations are presented for comparison.

2:34PM QX.00009 The Effect of Convex Sleeve on Cavitation Inception in the Rotary Valve of the Power Steering System1. GWI TAEK KIM, Seoul National University, SUN HONG PARK, MANDO Corp., MYUNG HWAN CHO, JUNG YUL YOO, Seoul National University — The rotary valve in power steering system helps the drivers turn the wheel with ease, when they set the vehicle in motion. It is well known that the hess noise in the rotary valve occurs by the cavitation effect due to high pressure drop at the orifice. In this paper, the flows in two types (Round and Straight types) of rotor valves have been analyzed numerically by using three-dimensional cavitation model embedded in the commercial code, FLUENT 12.0. The shapes of the sleeve grooves are convex and rectangular respectively in the Round and Straight types. The numerical results have been compared with the hess noise level measured in a semi-anechoic chamber. It is found that the shape of the sleeve grooves affects considerably the volume of the oil vapor generated from cavitation. These results can be utilized for the improved design of the hydraulic rotary valve with hess noise reduction.

1The authors are supported by the Korea Research Foundation Grant funded by the Korean Government (MOEHRD) (KRF-0420-2007005).

2:47PM QX.00010 Characterization of Turbulence and Cavitation Induced Pipe Vibrations Due to Flow thru Baffle Plates. GAVIN HOLT, DANIEL MAYNES, JONATHAN BLOTTER, Brigham Young University — We report experimental results characterizing pipe vibrations caused by turbulent flow and cavitation due to water flow through baffle plates mounted in a 10.2 cm diameter schedule 40 PVC pipe test section of a closed water flow loop. The baffle plates contained varying hole sizes that ranged from 0.159 cm to 2.54 cm, with the total through area, or openness, of each baffle plate ranging between 11% and 60%. Pipe wall acceleration data was collected for a range of Reynolds numbers from 5.85 x 10⁶. Acceleration measurements were acquired at locations along the pipe from 3-110 diameters downstream of the baffle plate. The measurements show that vibration levels at a given streamwise position increase with larger hole size, a decrease in openness, and increased flow rate. The incipient point of cavitation was observed to occur at decreasing flow rate with increasing hole size and decreasing openness. Vibration levels decreased asymptotically with increased distance downstream of the baffle plate for all scenarios and the streamwise distance at which the vibration level was attenuated increased as the hole size increased, openness decreased, and flow rate increased. The downstream vibration level also increased with these parameters.

Tuesday, November 23, 2010 12:50PM - 2:47PM Session QY.00001 Fingering Patterns of Ferrofluid Droplets In a Radial Field CHING-YAO CHEN, Logan, Utah, USA — Complex pattern formation abound in nature and has been actively studied in many different physical, chemical, and biological systems. One major point of interest is to understand the morphology of the rising patterns. In this context, the investigation of growth phenomena in ferrofluids has drawn considerable attention during the last few decades. Due to its unique response to applied magnetic fields, this fluid material has become a prototypical dipolar system for the study of a number of pattern-forming processes and interfacial instabilities. Pattern formation in a ferrofluid system under an in-plane radial magnetic field is experimentally investigated. Visually striking patterns are obtained. For miscible ferrofluids, the morphologies change from circular to a zero field to complex starburst-like structures at a finite field. Less vigorous fingering patterns evolve if the fluids are immiscible. The number of fingers can be tuned by applying a perpendicular field to introduce desired initial perturbation before the switching to in-plane field. The evolution of ferrofluid droplets of various initial diameters, subjected to different magnetic field strengths is considered to investigate their influences.

1:03PM QY.00002 On the breakup of nanoscale metallic rings melted via laser pulses. JAVIER DIEZ, Instituto de Física Arroyo Seco, Universidad Nacional del Centro de la Provincia de Buenos Aires, Pinto 399, 7000, Tandil, LOU KONDIC, Department of Mathematical Sciences and Center for Applied Mathematics and Statistics, New Jersey Institute of Technology, Newark, NJ, YUEYING WU, JASON FOWLKES, CHING-YAO CHEN, Logan, Utah, USA — We consider a pure liquid film with two liquid-gas interfaces — a free film — in two dimensions. Assuming that the aspect ratio of the film thickness to the arc length of the center-line is small, we develop a set of models using lubrication theory for the evolution of the film including the effects of different gas pressures above and below the liquid as well as strong surface tension. These models show a separation of timescales between center-line relaxation, thickness and drainage due to capillary suction from adjoining Plateau borders. The downstream vibration level also increased with these parameters.

1Supported by NSF RTG DMS-0636574 and CMMI-0826703.

1:16PM QY.00003 Models for metallic foam lamellae1. MICHAEL B. GRATTON, STEPHEN H. DAVIS, Northwestern University — We consider a pure liquid film with two liquid-gas interfaces — a free film — in two dimensions. Assuming that the aspect ratio of the film thickness to the arc length of the center-line is small, we develop a set of models using lubrication theory for the evolution of the film including the effects of different gas pressures above and below the liquid as well as strong surface tension. These models show a separation of timescales between center-line relaxation, thickness and drainage due to capillary suction from adjoining Plateau borders. The downstream vibration level also increased with these parameters.

1Supported by NSF RTG DMS-0636574 and CMMI-0826703.
usual inverse cubic law for the disjoining pressure, as in the classical case, slightly more general (non-singular) forms are also considered. In this situation, and shows that the apparent contact angle only weakly deviates from Young's law in that case. Finally, while most of this theory is based on the regime with a truncated microfilm, ending up at a bare surface. Consideration of this new regime requires introducing the spreading coefficient into the picture, extending all over the apparently dry portions of the superheated substrate.

We here show that such regime may actually become metastable against a new allowing the prediction of the apparent contact angle and of the microscale evaporation flux. The analysis is restricted to perfectly flat and homogeneous of evaporating contact lines is revisited (for the case of a one-component liquid and its pure vapor) in the framework of a continuum lubrication-type model allowing the prediction of the apparent contact angle and of the microscale evaporation flux. The analysis is restricted to perfectly flat and homogeneous substrates maintained at constant temperature. While the classical theory, used for perfectly wetting situations, assumes the existence of an adsorbed microfilm extending all over the apparently dry portions of the superheated substrate, we here show that such regime may actually become metastable against a new regime with a truncated microfilm, ending up at a bare surface. Consideration of this new regime requires introducing the spreading coefficient into the picture, hence in some sense unifying two apparently unrelated ways of modeling contact line microstructures. In particular, the analysis also applies to partial wetting situations, and shows that the apparent contact angle only weakly deviates from Young’s law in that case. Finally, while most of this theory is based on the usual inverse cubic law for the dispersing pressure, as in the classical case, slightly more general (non-singular) forms are also considered.

1:29PM QY.00004 A network model for foam dynamics

PETER STEWART, MICHAEL GRATTON, MICHAEL DAVIS, STEPHEN DAVIS, Northwestern University — We present a large-scale network model for the dynamics and stability of a planar metallic foam, composed of polygonal gas bubbles separated by thin liquid films. In particular, we track the positions of the bubble vertices, where most of the liquid volume is concentrated, and incorporate a direct coupling between the pressure and volume of the bubbles, surface-tension forces on the gas-liquid interfaces and draining and elongational flows in the films. We invoke a van-der-Waals instability criterion due to Anderson, Brush and Davis [to appear in J. Fluid Mech.] and present numerical simulations of the resulting topological re-arrangements within the foam.

1:42PM QY.00005 Electrically Driven Motion of Thin Films of Dielectric Liquids

PILNAM KIM, CAMILLE DUPRAT, Princeton University, SCOTT S.H. TSAI, Harvard University, HOWARD A. STONE, Princeton University, PRINCETON UNIVERSITY TEAM — In electrohydrodynamic (EHD) pumping, fluid forces are generated by the interaction of electric fields with the charges they induce in the fluid. Here, we investigate the effect of a tangential electric field on the motion of a thin film of a dielectric liquid in a wedge-shape geometry. We first present an experiment study. We find that the fluid is driven to high electrical potential regions due to the tangential field. In addition, the liquid interface undergoes an instability in the form of a conical shape (a Taylor cone) induced by a normal electric field to the interface. We propose a thin film model using lubrication theory to describe the film thickness-velocity relationship and characterize the jetting processes that accompany the interface instability.

1:55PM QY.00006 Viscous fingering of a miscible reactive A+B→C interface with an infinite Damkohler number: Nonlinear simulations

YUICHIRO NAGATSU, Nagoya Institute of Technology, Japan, ANNE DE WIT, NLPC, Universite Libre de Bruxelles, Belgium — Nonlinear simulations of miscible viscous fingering are performed for a reactive system where a simple infinitely fast A+B→C chemical reaction takes place when a solution containing the reactant A is displacing another miscible solution containing the reactant B. The viscosity of the fluid depends on the concentration of the chemicals B and C. The various nonlinear fingering dynamics are analyzed numerically for an infinite Damkohler number D∞ as a function of the log-mobility ratios Ro and Re quantifying the viscosity ratios of the solutions of B and C versus that of the solution of A respectively. If Ro ≫ 0, i.e. if the system is genuinely viscously unstable because the displaced solution of B is more viscous than the displacing solution of A, we analyze the changes to classical non-reactive viscous fingering induced by the reaction.

2:08PM QY.00007 Emulation of Mucus Propulsion in the Trachea Driven by Constant Air Flow

REED OGROSKY, ROBERTO CAMASSA, MICHAEL JENKINSON, JEFFREY OLANDER, SHREYAS TIKARE, UNC — To better understand the movement of mucus through the trachea that arises as a result of air flow, we design an experiment to emulate mucus movement by an air-driven vertical flow of high-viscosity silicone oil through a thin glass tube. When a constant flux of air is delivered through the bottom of the tube, instabilities arise, generating upward moving waves at the oil/air interface. These constitute a main mechanism of momentum transfer from air to oil, whereby oil is transported upward against gravity. We test this mechanism with several different flow rates of both air and oil. Specifically, increasing the air speed results in shorter wavelengths, lower wave speed, a smaller mean thickness of oil lining the tube, and smaller displacements by arriving waves at the wetting front when oil is advancing in a dry tube. In particular, we quantify the role of waves in advancing this front, and show how waves play a dominant role in this advancement. These results give insight into the clearing of mucus in the trachea by air flows.

2:21PM QY.00008 Moving contact lines in a vapor-liquid system: a singularity-free description in the sole framework of classical physics

ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles, TIPs-Fluid Physics — When one is lead to think about a theoretical treatment of moving contact lines in the sole framework of classical physics, the first associations coming to mind are most probably those of singularities intractable unless “regularizing” effects, beyond the classical approach, are taken into account, such as the disjoining pressure or a slip at the wall. Here we show that, contrary to such preconceptions, no contact-line singularities arise, even in the absence of these regularizing effects, in a system consisting of a liquid, its pure vapor and a superheated substrate (of interest, in particular, in boiling applications). Furthermore, no thermal singularities typically associated with this system are encountered either, even in the absence of the thermal regularizing effects such as a finite rate of the vaporization kinetics or a finite heat conductivity of the substrate. We consider, in the framework of the lubrication theory and a classical one-sided model, a contact line moving at a constant velocity (advancing or receding) and starting abruptly at a (formally) bare solid surface, the micro-contact angle being either equal to zero or finite.

3:34PM QY.00009 Steady contact lines in a vapor-liquid system: truncated versus extended adsorbed microfilms

PIERRE COLINET, ALEXEY REDNIKOV, Universite Libre de Bruxelles, TIPs laboratory — The classical microscale theory of evaporating contact lines is revisited (for the case of a one-component liquid and its pure vapor) in the framework of a continuum lubrication-type model allowing the prediction of the apparent contact angle and of the microscale evaporation flux. The analysis is restricted to perfectly flat and homogeneous substrates maintained at constant temperature. While the classical theory, used for perfectly wetting situations, assumes the existence of an adsorbed microfilm extending all over the apparently dry portions of the superheated substrate, we here show that such regime may actually become metastable against a new regime with a truncated microfilm, ending up at a bare surface. Consideration of this new regime requires introducing the spreading coefficient into the picture, hence in some sense unifying two apparently unrelated ways of modeling contact line microstructures. In particular, the analysis also applies to partial wetting situations, and shows that the apparent contact angle only weakly deviates from Young’s law in that case. Finally, while most of this theory is based on the usual inverse cubic law for the dispersing pressure, as in the classical case, slightly more general (non-singular) forms are also considered.

1:19PM QY.00010 Viscous fingering of a miscible reactive A+B→C interface with an infinite Damkohler number: Nonlinear simulations

1NIH

1Supported by ESA & BELSPO PRODEX projects and by FRS-FNRS.

1Supported by FRS-FNRS, and by ESA & BELSPO PRODEX projects.
12:50PM QZ.00001 Dynamics of a fluid inside a precessing cylinder. ROMAIN LAGRANGE, IFP, PATRICE MEUNIER, CHRISTOPHE ELOY, IRPHE, FRANCOIS NADAL, CEA, CNRS/CEA COLLABORATION — The instability of a fluid inside a precessing cylinder is studied theoretically and experimentally. This study is motivated by aeronautics and geophysics applications. Precessional motion forces hydromotions waves called Kelvin modes whose structure and amplitude are predicted by a linear inviscid theory. When a forced Kelvin mode is resonant, a viscous and weakly nonlinear theory has been developed to predict its saturated amplitude. We show that this amplitude scales as $\Re^{1/2}$ for low Reynolds numbers and as $\Re^{1/3}$ (where $\Re$ is the precessing angle) for high Reynolds numbers. These scalings are confirmed by PIV measurements. For Reynolds numbers sufficiently large, this forced flow becomes unstable. A linear stability analysis based on a triadic resonance between a forced Kelvin mode and two free modes has been carried out. The precessing angle for which the flow becomes unstable is predicted and compared successfully to experimental measurements. A weakly nonlinear theory was developed and allowed to show that the bifurcation of the instability of precession is subcritical. It also showed that, depending on the Reynolds number, the unstable flow can be steady or intermittent. Finally, this weakly nonlinear theory allowed to predict, with a good agreement with experiments, the mean flow in the cylinder, even if it is turbulent.

1:03PM QZ.00002 Inertial waves in the channel turbulence with system rotation1, YAN-TAO YANG, JIE-ZHI WU2, SKLTC, College of Engineering, Peking University, Beijing, 100871, China — We study inertial waves (IW) in the turbulent channel flow with system rotation about an axis in either streamwise or spanwise direction (STR or SPR). For the mean channel shear flow, we construct IW solutions by the helical wave functions (eigenfunctions of the curl operator), and show their existence when the basic flow satisfies certain conditions. These theoretical predictions were tested by DNS of the rotating channel flows. We have recently confirmed that the STR channel holds IWs when the rotating rate is high enough, and for different rotating rates the IWs have similar wavenumber and negative polarity (JFM, to appear). As for the SPR cases, the IWs may exist at even higher rotating rates than that needed for STR cases. They locate at the layer where the mean streamwise velocity is largest. It is likely that when a system rotation is applied to a unidirectional shear flow, the IWs may appear at the region where the mean streamwise velocity takes the local maximum. The profiles of the mean velocity are modified by IWs.

1:16PM QZ.00003 The motion and the forces on a cylinder and a sphere in rotating shear flow, YOSHIYUKI TAGAWA, CHAO SUN, The University of Twente, TOM MULLIN, Manchester Centre for Nonlinear Dynamics, LEEN VAN WULGAARDEN, DETLEF LOHSE, The University of Twente — The motion and rotation rate of a heavy cylinder and sphere inside a rotating drum were investigated. The drum, filled with distilled water, was rotating around its horizontal axis with varying rotation rates. The cylinder was observed to either co- or counter-rotate with respect to the rotating drum. The motion of the cylinder depends not only on the radius of the cylinder, but also on its length. The flow around the cylinder was measured with particle image velocimetry (PIV). Results on spheres with different radii will also be presented.

1:29PM QZ.00004 Viscous spreading of an inertial wave beam in a rotating fluid. PIERRE-PHILIPPE CORTET, CYRIL LAMRIBEN, FREDERIC MOISY, Laboratoire FAST, CNRS UMR 7608, Universite Paris-Sud, Universite Pierre-et-Marie-Curie, ROTATING TURBULENCE TEAM — We report experimental measurements of inertial waves generated by an oscillating cylinder in a rotating fluid. The two-dimensional wave takes place in a stationary cross-shaped wavepacket. Velocity and vorticity fields in a vertical plane normal to the waveraker are measured by a corotating Particle Image Velocimetry system. The viscous spreading of the wave beam and the associated decay of the velocity and vorticity envelopes are characterized. They are found in good agreement with the similarity solution of a linear viscous theory, derived under a quasi-parallel assumption similar to the classical analysis of Thomas and Stevenson [J. Fluid Mech. 54 (3), 495–506 (1972)] for internal waves.

1:42PM QZ.00005 Spin-Up and Spin-Down in a Half Cone, MICHAEL PATTERSON, University of Bath, LIGANG LI, KEKE ZANG, University of Exeter, RICH KERSWELL, University of Bristol — The spin-up and spin-down flow responses in a rapidly-rotating, fluid-filled, closed half-cone are studied both numerically and experimentally. This unusual set up is of interest because it represents a pathological case for the classical linear theory of Greenspan & Howard (1963) since there are no closed geostrophic contours nor a denumerable set of inertial waves. Yet, the flows observed are surprisingly simple except when the fluid is appreciably spun-down which induces boundary layer separation and complicated spatiotemporal behaviour. Most notably, the linear regime of small increase or decrease in the rotational speed exhibits the familiar “spin-up” Ekman boundary layer timescale of $O(\Re^{1/2})$ (where $E$ is the Ekman number) for adjustment.

1:55PM QZ.00006 Tilt-over mode in a precessing triaxial ellipsoid. DAVID CEBRON, MICHAEL LE BARS, PATRICE MEUNIER, IRPHE — The tilt-over mode in a precessing triaxial ellipsoid is studied theoretically and numerically. Inviscid and viscous analytical models previously developed for the spheroidal geometry by Poincare (1910) and Busse (1968) are extended to this more complex geometry, which corresponds to a tidally deformed spinning astrophysical body. As confirmed by three-dimensional numerical simulations, the proposed analytical model provides an accurate description of the stationary flow in an arbitrary triaxial ellipsoid, until the appearance at more vigorous forcing of time dependent flows driven by tidal and/or precessional instabilities.

2:08PM QZ.00007 Experimental Determination of Zonal Winds Driven by Tides, CYPRIEN MORIZE, FAST, MICHAEL LE BARS, PATRICE LE GAL, IRPHE, ANDREAS TILGNER, University of Göttingen, MORIZE COLLABORATION, LE BARS, LE GAL TEAM, TILGNER COLLABORATION — We describe a new phenomenon of zonal wind generation by tidal forcing. Following a recent theoretical and numerical analysis [A. Tilgner, Phys. Rev. Lett. 99, 194501 (2007)], we present the first experimental evidence that the nonlinear self-interaction of a tidally forced inertial wave (IW) can drive an intense axisymmetric flow in a rotating deformed sphere. Systematic measurements of zonal flows are carried out by an embarked instrument on a rotating drum, filled with distilled water, was rotating around its horizontal axis with varying rotation rates. The cylinder was observed to either co- or counter-rotate with respect to the rotating drum. The motion of the cylinder depends not only on the radius of the cylinder, but also on its length. The flow around the cylinder was measured with particle image velocimetry (PIV). Results on spheres with different radii will also be presented.

2:21PM QZ.00008 Lagrangian velocity, acceleration and vorticity autocorrelations in rotating turbulence. HERMAN J.H. CLERCX, LORENZO DEL CASTELLO, Eindhoven University of Technology — The influence of the Earth background rotation on oceanic and atmospheric currents, as well as the effects of a rapid rotation on the flow inside industrial machineries like mixers, turbines, and compressors, are typical examples of fluid flows affected by rotation. Rotating turbulence has often been studied by means of numerical simulations and analytical models, but the experimental data available is scarce and purely of Eulerian nature. In the present study, experiments on continuously forced turbulence subjected to different background rotation rates are performed by means of 3D Particle Tracking Velocimetry. The data collected is processed in the Lagrangian frame, as well as in the Eulerian one. The background rotation is confirmed to induce 2-dimensionalisation of the velocity field, and the large-scales are dominated by stable counter-rotating vertical tubes of vorticity. The auto- correlation coefficients along particle trajectories of velocity, acceleration and vorticity components have been explored, and in this talk the effects of rotation on the Lagrangian temporal scales of the flow will be discussed.
2:34PM QZ.00009 A closed grid turbulence experiment under rotation: Anisotropic energy transfer and inertial modes inhibition. CYRIL LAMRIBEN, PIERRE-PHILIPPE CORTEZ, FREDERIC MOISY, Laboratoire FAST, CNRS UMR 7608, Université Paris-Sud, Université Pierre-et-Marie-Curie, ROTATING TURBULENCE TEAM — We report an experimental study of the free decay of an initially 3D homogeneous and isotropic grid turbulence submitted to a global rotation. Turbulence is generated by rapidly towing a grid in a rotating water tank and velocity fields are measured in a vertical plane parallel to the rotation axis using a corotating Particle Image Velocimetry (PIV) system. We first show that, when a simple grid is used, a significant amount of the kinetic energy is stored in a reproducible mean flow composed of resonant inertial modes and whose spatial structures are extracted. The possible coupling between these modes and turbulence suggests that turbulence cannot be considered as freely decaying. We demonstrate that these inertial modes may be considerably inhibited by adding inner tanks to the grid, yielding, for the first time, an effectively freely decaying rotating turbulence in a confined geometry. We also provide a thorough analysis of the anisotropic energy transfers from the anisotropic third order moment of the velocity increments obtained by PIV. We show that the departure from 3D isotropic energy transfers is stronger for horizontal increments. We also show a direct evidence of an inverse energy cascade of horizontal motions at large scales.

2:47PM QZ.00010 Vortex Induced Flows Involving Fused Cylinder/Sphere Pairs. D. PALANIAPPAN, Texas A&M University Corpus Christi — The two-dimensional inviscid flow field around an infinitely long circular cylinder induced by a point vortex can be constructed via the famous Milne-Thomson’s circle theorem. However, for non-circular configurations such calculations become cumbersome and even the computation of approximate solutions of the Euler equations governing the fluid flow require quite a lot of guess work. Here we show a systematic procedure to derive analytic results for a geometry consisting of two overlapping cylinder/sphere pairs in the limit of inviscid flow. Our small approach yields the image system, also known as Neumann’s Green function, describing the complete flow field induced by a vortex in the presence of a twin-circle configuration. The exact results for the uniform flow-vortex-twin circle combination are obtained via our successive-image theory and are expressed in the form of Hamiltonian/streamfunction for the system. The flow topologies for this system reveal the existence of stagnation points and streamline crossing, a strong indication of chaos. The corresponding analytic solutions for two fused spheres submerged in a flow induced by a three-dimensional vortex are also obtained. The exact solution for the twin-sphere problem is expressed in terms of standard convergent elliptic integrals which can be evaluated numerically. The results of this investigation, in general, illustrate the topography effects and are of fundamental importance in oceanography and other related topics as well.

Tuesday, November 23, 2010 3:05PM - 4:36PM – Session RA Turbulence Theory III Long Beach Convention Center 101A

3:05PM RA.00001 Effect of scalar-field boundary conditions on the Markovian properties of passive scalar increments. JASON LEPORRE, LAURENT MYDLARSKI, McGill University — Lepore and Mydlarski recently investigated the influence of the scalar-field boundary conditions on the inertial-convective-range scaling exponents of the high-order passive scalar structure functions \( \xi_n \). The latter was accomplished by injecting the scalar field into the flow (i.e., the turbulent wake of a circular cylinder) using two different scalar injection methods: (i) heating the cylinder, and (ii) using a "mandoline". The authors concluded that all previous estimates of \( \xi_n \) are sensitive to the scalar field boundary conditions, given the finite Reynolds numbers of the flows under consideration, and, therefore, do not constitute a universal measure of the internal intermittency of the passive scalar field. The present work examines the Markovian properties of passive scalar increments, and their dependence on the scalar injection method, to provide additional insight into the small-scale structure of the turbulent passive scalar. In particular, the current research examines the relationship between the high-order terms of the Kramers-Moyal expansion and the internal intermittency of the passive scalar field.

3:18PM RA.00002 Tomographic Particle-Image Velocimetry To Investigate Dissipation Elements. LISA SCHAIFER, Institute of Aerodynamics, RWTH Aachen University, UWE DIERKSHEDER, LaVision GmbH, WOLFGANG SCHROEDER, Institute of Aerodynamics, RWTH Aachen University — A new method to describe the nature of turbulence has been proposed by Wang and Peters (JFM 2006). Based on fluctuating scalar fields, local minimum and maximum points are determined via gradient trajectories starting from every grid point in the direction of the steepest ascending and descending gradients. Then, so-called dissipation elements are identified as the region of all the grid points the trajectories of which share the same pair of minimum and maximum points. The statistical properties of these space-filling elements are evaluated focusing on the linear distance and the scalar difference between their extrema. The procedure is also applied to various DNS fields using \( u^{'}, v^{'}, w^{'}, \text{ and } k^{' } \) as scalar fields (Wang and Peters JFM 2008). In this spirit, dissipation elements are derived from experimental 3D velocity data of a fully developed turbulent channel flow gained by Tomographic Particle-Image Velocimetry. The statistical results, inter alia, regarding the distribution of the element length are compared to those from the DNS.

3:31PM RA.00003 Forced Turbulence, Multiscale Dynamics, and Variational Principles. HARIS J. CATRAKIS, University of California, Irvine — We consider theoretically fundamental aspects of forced turbulence as well as unforced turbulence, with emphasis on the multiscale properties of turbulent level crossings as well as emphasis on connections to variational principles. The connection between power spectral exponents and level crossing scales in forced turbulence, as well as unforced turbulence, is explored. Also, the connection between variational principles and the behavior of level crossing scales is investigated in both forced and unforced turbulence. In addition, we explore testing of our theoretical considerations using computations and visualizations.

3:44PM RA.00004 Recent Analytical and Numerical Results for The Navier-Stokes-Voigt Model and Related Models. ADAM LARIOS, University of California Irvine, Los Alamos National Lab, EDRISS TITI, University of California Irvine, Weizmann Institute of Science, MARK PETERSEN, BETH WINGATE, Los Alamos National Lab — The equations which govern the motions of fluids are notoriously difficult to handle both mathematically and computationally. Recently, a new approach to these equations, known as the Voigt-regularization, has been investigated as both a numerical and analytical regularization for the 3D Navier-Stokes equations, the Euler equations, and related fluid models. This invalid regularization is related to the alpha-models of turbulent flow; however, it overcomes many of the problems present in those models. I will discuss recent work on the Voigt-regularization, as well as a new criterion for the finite-time blow-up of the Euler equations based on their Voigt-regularization. Time permitting, I will discuss some numerical results, as well as applications of this technique to the Magnetohydrodynamic (MHD) equations and various equations of ocean dynamics.
3:57PM RA.00005 Is drift-wave turbulence intermittent from a Lagrangian point of view? KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, France, BENJAMIN KADOCHE1, M2P2-CNRS & Ecole Centrale de Marseille, Marseille, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon - Universite de Lyon, Ecully, France — Lagrangian velocity statistics of dissipative drift-wave turbulence are investigated by means of direct numerical simulation in the context of the Hasegawa-Wakatani model. For large values of the adiabaticity (or small collisionality), the probability density function of the Lagrangian acceleration shows exponential tails, as opposed to the stretched exponential or algebraic tails, generally observed for the highly intermittent acceleration of Navier-Stokes turbulence. This exponential distribution is shown to be a robust feature independent of the Reynolds number. For small adiabaticity, algebraic tails are observed, suggesting the strong influence of point-vortex-like dynamics on the acceleration. A causal connection is found between the shape of the probability density function and the auto-correlation of the norm of the acceleration. For further details we refer to Bos et al., Physica D 239, 2010 and Kadoch et al., Phys. Rev. Lett., 2010, in press.

4:10PM RA.00006 Maximum Enstrophy Growth in Burgers Equation, DIEGO AYALA, BARTOSZ PROTAS, Department of Mathematics and Statistics, McMaster University — The regularity of solutions of the Navier–Stokes equation is controlled by the boundedness of the enstrophy $E$. The best estimate for its rate of growth is $dE/dt \leq C E^3$, for $C > 0$, leading to the possibility of a finite–time blow–up when straightforward time integration is used. Recent numerical evidence by Lu & Doering (2008) supports the sharpness of the instantaneous estimate. Thus, the central question is how to extend the instantaneous estimate to a finite–time estimate in a way that will incorporate the dynamics imposed by the PDE. We state the problem of saturation of finite–time estimates for the enstrophy growth as a PDE–constrained optimization problem, using the Burgers equation as a “toy model”. The following problem is solved numerically:

$$\max \{E(T) - E(0)\} \text{ subject to } E(0) = E_0$$

where $\phi$ represents the initial data for Burgers equation, for a wide range of values of $T > 0$ and $E_0$ finding that the maximum enstrophy growth in finite time scales as $E_0^3$ with $\alpha \approx 3/2$, an exponent different from $\alpha = 3$ obtained by analytic means.

4:23PM RA.00007 Turbulence in more than two and less than three dimensions, ANTONIO CELANI, Institut Pasteur Paris, DARIO VINCENZI, STEFANO MUSACCHIO — We investigate the behavior of turbulent systems in geometries with one compactified dimension. A novel phenomenological scenario dominated by the splitting of the turbulent cascade emerges both from the theoretical analysis of passive scalar turbulence and from direct numerical simulations of Navier-Stokes turbulence. (Phys. Rev. Lett. 104, 184506 [2010], J. Stat. Phys. 138, 579-597 [2010]).

Tuesday, November 23, 2010 3:05PM - 4:49PM
Session RB Turbulence Simulations VI Long Beach Convention Center 101B

3:05PM RB.00001 Geometric nature of particle trajectory in isotropic turbulence, YONGNAM PARK, Yonsei University, YEONTAEK CHOI, National Institute for Mathematical Science, CHANGHOON LEE, Yonsei University — The geometric nature of particle trajectory is investigated for understanding the Lagrangian nature of turbulence using direct numerical simulation of isotropic turbulence. Probability density functions and autocorrelations along a fluid particle trajectory associated with geometric quantities such as curvature and torsion of the Lagrangian trajectory are provided. We propose the ratio of torsion to curvature as an important parameter to identify the particle trajectory, and it is found to play a crucial role in understanding the geometric shape of particle trajectory. The relationship between Lagrangian helicity and the ratio of torsion to curvature is investigated where Lagrangian helicity is defined as a dot product of velocity and vorticity vectors at the point of a fluid particle. We also found that probability density functions of torsion and torsion normalized by curvature clearly show well-established slope in log–log plots. Lagrangian helicity is intermittently distributed and high Lagrangian helicity is always found, where high acceleration is observed. Regarding the relationship between coherent structure and acceleration, coherent structure can be understood in terms of Lagrangian helicity, curvature, and torsion. Geometric characteristics for solid particles are also investigated and its behavior differs depending on the Stokes number.

3:18PM RB.00002 Turbulence computations on a 4096^3 periodic domain: passive scalars at high Schmidt number and Lagrangian statistics conditioned on local flow structure1, PUL-KUEN YEUNG, Georgia Tech, D.A. DONZIS, Texas A&M Univ., K.R. SREENIVASAN, New York Univ., B.L. SAIVFORD, Monash Univ., Australia, S.B. POPE, Cornell Univ. — Rapid advances in Cyberinfrastructure, with more to come on the horizon, are presenting many opportunities for extending simulations of turbulence towards previously inaccessible parameter regimes and improved results in problems with greater complexity. In our group we have performed 4096^3 simulations of isotropic turbulence on three massively parallel machines to study turbulence at higher Reynolds number, higher Schmidt number, or better resolution than previously accessible. One topic studied is Batchelor scaling and small-scale intermittency of passive scalar fields in turbulent mixing at high Schmidt number with a demonstrable viscous-convective range. Another is the behavior of Lagrangian structure functions at high Reynolds number ($R_i \approx 1000$), with conditional sampling used to distinguish between the characteristics of strain-dominated versus rotation-dominated regions of the flow. We shall discuss both problems briefly, and conclude with an overview of current and future challenges involved in striving towards the next level, involving Petascale computing and beyond.

3:31PM RB.00003 Behavior of Heavy Particles in Turbulent Channel Flow, JUNGHOO LEE, CHANGHOON LEE, Yonsei University — The motion of heavy particles in turbulent channel flow was investigated by using direct numerical simulation. We assumed that Stokes drag, Saffman lift and Magnus lift act on the motion of heavy spherical particles in turbulence. In this study, Stokes number is defined as the particle response time normalized by the wall units. The range of the Stokes number is 0.1~50 and the diameter of a particle is 0.06~3 in wall unit. Collision of particles with the wall is modelled by an elastic collision. Relevant velocity and acceleration statistics of heavy particles for the given range of Stokes number were investigated to interpret the particle accumulation near the wall. Particle accumulation at the wall is maximized when the Stokes number is around 15. And we found that Saffman lift force has a great effect on particle acceleration in the wall-normal direction near the wall. Detailed statistics including probability density function and autocorrelation of particle velocity and acceleration will be presented in the meeting.

1Supported by NSF Grants CBET-0553867 and OCI-0749223.
3:44PM RB.00004 Maximum Drag Reduction Asymptote in Turbulent Channel Flow of Polymer Solutions. CHANG-FENG LI, Jiangsu University, P.R. China, RACHAKRISHNA SURESHKUMAR, Syracuse University, BAMIN KHOMAMI, University of Tennessee, Knoxville — It is well known that the addition of a small amount of soluble high molecular weight polymers to wall bounded turbulent flows can lead to dramatic drag reduction (DR). Salient features of this phenomenon include: (1) existence of threshold for the onset of DR, and (2) an upper bound referred to as the maximum drag reduction (MDR) or the Virk asymptote. Computational studies including DNS and viscoelastic exact coherent structures have provided significant insight into the mechanism by which polymers alter turbulence and give rise to DR. Despite the significant progress in understanding polymer induced drag reduction in the low (DR<40%) and high (40%<DR<60%) DR regimes, fundamental understanding of existence of a universal upper limit of drag reduction and the nature of the flow at this limit is still lacking. In this study, we have developed new mechanistic insight at MDR both in terms of the existence of a universal upper limit and the nature of the flow in this regime by analyzing extensive high-fidelity direct numerical simulation data of turbulent channel flows of dilute polymeric solutions.

3:57PM RB.00005 Normal stress difference and drag reduction mechanism in Johnson-Segalman viscoelastic turbulence. KIYOSI HORIUTI, Tokyo Institute of Technology, Japan, KAZUMA MATSUMOTO — The mechanism of turbulent drag reduction in the polymer-diluted flow is studied using the DNS data for homogeneous isotropic turbulence and pipe flow. The polymer stress $\tau$ is obtained by solving the non-affine Johnson-Segalman constitutive equation. The drag reduction is maximal when non-affinity is either minimum or maximum, but the largest reduction is achieved when non-affinity is maximum. The pressure force due to $\tau$, $\nabla p$, tends to oppose to that due to the solvent $\nabla p_s$, e.g., in the core of the vortex tube in which $p_s$ is minimal, $p_s$ bulges out. The normal-stress difference (NSD) is obtained on the basis of new eigenvectors which span the isosurfaces of vortex tube and sheet. It is shown in both flows that the first NSD is predominantly positive and the second is negative along the sheets and tubes. Thus, an extra tension is exerted on the sheet and tube. With an increase of effective viscosity by an addition of elongation viscosity, resistance of the sheet and tube to their stretching is enhanced. When non-affinity is maximum, the transformation of the sheet into the tube is restrained because the sheet tends to snap back to the original flat form due to viscoelastic effect. When non-affinity is minimum, the tubes are created but its stretching is suppressed. In both cases, cascade of the energy into the small scales is restricted leading to the reduction of drag.

4:10PM RB.00006 Statistics of polymer extensions in turbulent channel flow. F. BAGHERI, Linne Flow Centre, KTH Mechanics, Sweden, D. MITRA, NORDITA, Sweden, P. PERLEKAR, Eindhoven University of Technology, The Netherlands, L. BRANDT, Linne Flow Centre, KTH Mechanics, Sweden — We carry out direct numerical simulations of three dimensional channel flow with passive polymer additives. We also calculate, for the first time, the PDF of finite-time Lyapunov exponents and from them the corresponding Cramer’s function for the channel flow. We study the statistics of polymer elongation for both the Oldroyd-B model (for Wi less than 1) and the FENE model. We use the location of the minima of the Cramer’s function to define the Weissenberg number precisely such that we observe coil-stretch transition at Wi approximately 1. For the Oldroyd-B model we find that the PDF of polymer extensions shows power-law behavior irrespective of the wall-normal coordinate of the polymer molecule, but the range of scaling does depend on the wall-normal coordinate. The exponent of this power-law matches with the earlier theoretical results within error bars. In addition we also find the dependence of the PDF of polymer extension on the wall-normal coordinate, v.i.z, the polymer are more stretched near the wall than at the center of the flow. We further study the orientation of the polymers with respect to the channel geometry. Our results show that the polymers close to the wall have a very high probability of being oriented along the stream-wise direction of the flow.

4:23PM RB.00007 Effect of inter-particle collision in particle-laden homogeneous isotropic turbulence. OHJOON KWON, CHANGHOON LEE, Yonsei University — It has been known that inter-particle collision of small spherical particles in particle laden homogeneous isotropic turbulence modifies the statistical characteristics of particle behaviour. For example, the dispersion of heavy particles is decreased by this collision effect (Lavieville 1997). Still, the mechanism of interaction between particle’s collision and turbulence is not clear. Direct numerical simulations are performed for particle-laden isotropic turbulence by adopting a spectral method, and the 4th-order Hermite interpolation is used for tracking particles. Inter-particle collision is considered as a complete energy conserved elastic process. Because the collision-induced particle acceleration is not quite related with the particle’s current-state properties and this irrelevance becomes dominant as the Stokes number increases, there are some significant statistical modifications observed in the behaviour of high Stokes number particles. We found that the particle velocity integral time scale decreases as the Stokes number becomes larger due to enhanced interference due to collision. Furthermore, the particle dispersion is suppressed for the same reason. The particle concentration tendency is also slightly mitigated due to the presence of particle volume and by collision. But there is no clear evidence about the variation of particle velocity variance. More detailed results will be presented in the meeting.

4:36PM RB.00008 Direct numerical simulation of strained turbulent flows and particles within. CHUNG-MIN LEE, California State University Long Beach, PRASAD PERLEKAR, FEDERICO TOSCHI, Technische Universiteit Eindhoven, ARMAN GYLFASON, Reykjavik University — We present results from direct numerical simulations of strained turbulent flows. Our focus is on the influence of the straining on the motions of passive and inertial particles of varied Stokes numbers. The results are compared with existing numerical and experimental data, and we seek to emphasize the effects of strain geometry and strain rate on the particle behavior. Eulerian flow field results, and the Lagrangian particle velocity and acceleration statistics are presented. The Rogallo algorithm is applied for simulating the flow field in a non-cubical domain.

Tuesday, November 23, 2010 3:05PM - 4:49PM – Session RC Rarefied Gases and Direct Simulation Monte Carlo — Long Beach Convention Center 102A

30:05PM RC.00001 Measures of Thermal Transpiration Flow. MARCOS ROJAS, IRINA GRAUR, PIERRE PERRIER, I.U.S.T.I. UMR 6595, Ecole Polytechnique Universitaire de Marseille — Thermal transpiration is the macroscopic movement of gas-particles induced by a temperature gradient. The gas-particles move from the lower to the higher temperature zone. The main aim of the present work is to measure experimentally the flow created by thermal transpiration in a tube heated on its outlet. The experimental system is composed by a circular cross section micro-tube and two reservoirs settled respectively at the inlet and outlet of the capillary. The reservoirs are coupled to two high-speed response time capacitance diaphragm gauges which monitor the pressure variation in time. By monitoring the pressure variation in time is possible to measure the macroscopic movement of gas-particles along the tube: by thermal transpiration gas-particles move from the cold to the hot region of the tube increasing the hot-side reservoir absolute pressure; while a drop of pressure is registered in the cold-side reservoir. The experiments are conducted for three different gases, Argon, Helium and Nitrogen, in a pressure range from 0.1 to 10 torr and for three different temperature differences: $\Delta T = 30, 40, 50$ degrees. The gas rarefaction conditions go from transitional to slip regime.

1This research is funded by the [European Community’s] Seventh Framework Programme ([FP7/2007-2013] under grant agreement n-215504
3:18PM RC.00002 Plane Poiseuille Flow of a Rarefied Gas in the Presence of a Strong Gravitation. TOSHIIKU DOI, Tottori University — Poiseuille flow of a rarefied gas between two horizontal planes in the presence of a strong gravitation is considered, where the gravity is so strong that the path of a molecule is curved considerably as it ascends or descends the distance of the planes. The gas behavior is studied based on the Boltzmann equation. An asymptotic analysis for a slow variation in the longitudinal direction is carried out and the problem is reduced to a spatially one dimensional problem, as was in the Poiseuille flow problem in the absence of the gravitation. The mass flow rate as well as the macroscopic variables is obtained for a wide range of the mean free path of the gas and the gravity. A numerical analysis of a two dimensional problem is also carried out and the result of the asymptotic analysis is verified.

3:31PM RC.00003 Surface Effects on Nanoscale Gas Flows. ALI BESKOK, MURAT BARIŞIK, Old Dominion University — 3D MD simulations of linear Couette flow of argon gas confined within nano-scale channels are performed in the slip, transition and free molecular flow regimes. The velocity and density profiles show deviations from the kinetic theory based predictions in the near wall region that typically extends three molecular diameters (s) from each surface. Utilizing the Irwin-Kirkwood theorem, stress tensor components for argon gas confined in nano-channels are investigated. Outside the 3s region, three normal stress components are identical, and equal to pressure predicted using the ideal gas law, while the shear stress is a constant. Within the 3s region, the normal stresses become anisotropic and the shear stress shows deviations from its bulk value due to the surface virial effects. Utilizing the kinetic theory and MD predictions for shear stress values, the tangential momentum accommodation coefficient for argon gas interacting with no FCC structured walls (100) plane facing the fluid is calculated to be 0.75; this value is independent of the Knudsen number. Results show emergence of the 3s region as an additional characteristic length scale in nano-confined gas flows.

3:44PM RC.00004 High Knudsen Number Fluid Flow at Near-Standard Temperature and Pressure Conditions using Precision Nanochannels. SUBHRA DATTA, A.T. CONLISK, The Ohio State University, WILLIAM H. FISSELL, Cleveland Clinic, SHUVO ROY, University of California, San Francisco, JEFF MAJESTRELLI, Cleveland Clinic — Gas flows over a wide range of Knudsen numbers (~0.5-10) are studied using silicon nano-channel arrays with slit-shaped pores that range from micrometer to sub-10nm scales. The flows are generated under conditions of room temperature and near-atmospheric pressure (~22°C and ~101-115 kPa) and span the continuum flow, continuum-slip flow, transition flow and free-molecular flow regimes. The measured flow rates of helium, argon and carbon dioxide are in good agreement with the Unified Slip Model (USM) proposed by Beskok and Karniadakis (Beskok A., Karniadakis G.E., Nanoscale and Microscale Thermophysical Engineering 3 (1999), no. 1, 43-77). The measured volumetric gas flow rates well agree with calculations based on the USM up to a Knudsen number of about Kn ~ 4, well into the transition regime; above this value the agreement for much of the data is qualitative and at very large Knudsen numbers the data is in the free molecular regime as expected.

3:57PM RC.00005 Energy accommodation of gas molecules with free-standing vertically aligned single-walled carbon nanotube arrays. IKUYA KINEFUCHI, KIZEN RYU, KEI ISHIKAWA, JUNICHIRO SHIOMI, SHU TAKAGI, SHIGEO MARUYUJI, YOICHIRO MATSUMOTO, The University of Tokyo — The scattering process of gas molecules on vertically aligned single-walled carbon nanotubes (VA-SWNTs) was investigated by molecular beam technique. The measurement was performed for the free-standing samples, which enabled us to evaluate the scattering process of gas molecules on VA-SWNT films themselves in detail without the presence of substrates. The scattered molecules are divided into three components: reflected molecules, diffusively transmitted molecules, and directly transmitted molecules without interaction with SWNTs. Even for the film thickness as thin as 0.1 μm, the incident molecules are found to be well accommodated to the surface temperature. This result suggests that, regardless of film thickness, most molecules have enough number of collisions with SWNTs for efficient energy transfer at the randomly oriented layer at the topmost of the films.

4:10PM RC.00006 Numerical Analysis on Energy Accommodation Process of Gas Molecules on Carbon Nanotube Film. JUMPEI KAWASAKI, IKUYA KINEFUCHI, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — Because of its large specific surface area due to its nanoscale structure, film made up of trapping gas molecules and enhancing heat exchange. The relationship between the diffusive movement of gas molecules inside the film and the energy accommodation process, however, has not been clarified yet. In this study, we introduced numerical simulation to analyze the energy accommodation process of gas molecules on CNT film. The film was modeled by using the piling up cylinders representing CNT bundles. Different scattering models between a CNT and a gas molecule were used to investigate the scattering angle distributions, number of collisions, and intrusion depths of gas molecules that were reflected by and transmitted into the film. The results of calculations well reproduced experimental results. We confirmed that although energy exchange of each collision between single CNT and a gas molecule is small, randomly oriented structure of CNT film induced diffusive movement of gas molecules inside the film, which leads to high accommodation between the film and gas molecules.

4:23PM RC.00007 Nonlinear and coupling effects on the gaseous Knudsen layer near the solid surface. RHO SHIN MYONG, Gyeongsang National University — The Knudsen layer (also known as the kinetic boundary layer) found in the region of a gas flow very close to the solid surfaces plays a critical role in modeling the rarefied and micro-scale gas flows. Although the Knudsen layer has been investigated extensively using kinetic theory in the past, capturing it within the continuum framework, which may provide distinct advantages in terms of computational efficiency, remains a daunting task. In particular, the exact underlying mechanisms behind abnormal behaviors in the Knudsen layer (smaller velocity slip and shear stress, nonlinear velocity profile, velocity gradient singularity, non-zero tangential heat flux) are not understood fully. In this work, those questions associated with the nonlinear and coupling effects in shear stresses and heat fluxes are investigated on the basis of the phenomenological nonlinear coupled constitutive relation (NCCR) and gas-surface molecular interaction model.

4:36PM RC.00008 The reality of diffuse volume transport. HOWARD BRENNER, MIT — Whereas mass flow in a continuum is necessarily accomplished by a concomitant volume flow, the converse is not true. That is, volume can be transported independently of mass. In particular, volume can be transported by purely molecular or diffuse action. For example, volume unaccompanied by mass can flow through a fluid that is completely at rest (and hence for which there is no mass flow) provided that density gradients exist. This occurs, for example, during steady-state heat conduction in quiescent isobaric liquids and gases, wherein the density varies locally with the temperature. Equally striking is the fact that volume can be transported through solid walls impermeable to matter. This talk is devoted to analyzing a gedanken experiment permitting these phenomena to be directly visualized, and hence providing indisputable evidence that diffuse volume transport is a real physical phenomenon relevant to continuum fluid mechanics.

Tuesday, November 23, 2010 3:05PM - 4:36PM Session RD Turbulence Simulations VII Long Beach Convention Center 102B
3:05PM RD.00001 Simulations of the wake of an accelerating body , MATTHEW DE STADLER, SUTANU SARKAR, University of California San Diego — When a body moving under its own power maneuvers, momentum is transferred to the surrounding fluid. This transfer of momentum, even for relatively small values, is thought to significantly affect the wake dynamics. Direct Numerical Simulation (DNS) in a spatially evolving frame, a first study of this kind, in addition to the commonly used temporal approximation, was used to evaluate this hypothesis. The main objective is to characterize the time evolution of the self-propelled wake dynamics in the presence of a momentum imbalance in a stratified fluid. Statistics of interest include the defect velocity, wake dimensions, vortex dynamics, the presence or lack thereof of large eddies in the late wake, wake lifetime, internal wave dynamics and mean and turbulent kinetic energies and their associated budgets. Results from the spatially evolving case are compared with data from a temporally evolving case with excess momentum.

3:18PM RD.00002 Turbulent Flow Past Projectiles: A Computational Investigation1 , IGIL MEHMEDEVIC, DONALD CARLUCCI, LIAM BUCKLEY, PASQUALE CARLUCCI, U. S. Army, ARDEC, Picatinny Arsenal, SIVA THANGAM, Stevens Institute of Technology, ARDEC-STEVENS COLLABORATION — Projectiles with free spining bases are often used for smart munitions to provide effective control, stability and terminal guidance. Computer results and investigations are performed for five case cylinders aligned along their axis where a base freely spins while attached to and separated at various distances from a non-spinning fore-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 and energy along with the modeled form of transport equations for the turbulence kinetic energy and the scalar form of turbulence dissipation. Computations are performed for both rigid cylinders as well as cylinders with free spining bases. Experimental data for a range of spin rates and free stream flow conditions obtained from subsonic wind tunnel with sting-mounted spinning cylinders is used for validating the computational findings.

3:31PM RD.00003 Direct Numerical Simulation of a Film Cooling Configuration with a Micro-ramp Vortex Generator1 , KAN WANG, MENG WANG, University of Notre Dame — Large-eddy simulations are carried out for compressible flow over a cylindrical turbine with a flat optical window to study the aero-optical distortions and their mitigation by passive control devices in the upstream boundary layer. The control devices consist of long and thin pins as in an experiment conducted at the University of Notre Dame. A comparison with the baseline case without pins shows overall agreement in terms of velocity statistics and the optical distortion magnitude. The root-mean-square of optical path difference (OPD) caused by the separated shear layer above the optical window is found to be five times as large as that caused by the attached boundary layer upstream of the turret. Simulation results for the passive-control case confirm key experimental observations. A second shear layer above the main shear layer is observed, which reduces the turbulence intensity of the main shear layer and widens the turbulence region over the optical window. The combined effect of the two shear layers leads to slightly reduced optical distortions compared with the uncontrolled flow with a single strong shear layer. Control strategies for reducing optical distortions without suppressing flow separation will be discussed.

3:44PM RD.00004 Application of Immersed Boundary Method to DNS of Stratified Flows , NARSIMHA RAPAKA, SUTANU SARKAR, University of California San Diego — An immersed boundary method is implemented in a Navier-Stokes solver that uses a mixed RK3-ADI time integration scheme with the viscous terms treated implicitly. A predictor-corrector algorithm is used to solve the momentum equations on a collocated grid arrangement. Simulations are performed for flow past a sphere and the results, including separation angle, vortex core position and the Strouhal frequency, agree closely with the literature. The present focus is on the numerical behavior of the solver in problems involving nonlinear internal tides on a model topography. Physical quantities of interest include the turbulent kinetic energy, turbulent dissipation rate, buoyancy flux as well as the energy flux and spectra associated with the propagating internal waves.

3:57PM RD.00005 Direct Numerical Simulation of a Film Cooling Configuration with a Micro-ramp Vortex Generator1 , AARON SHINN, S. PRATAP VANKA, University of Illinois at Urbana-Champaign — A Direct Numerical Simulation (DNS) of an inclined turbulent jet interacting with a cross-flow in a film cooling configuration is performed. The inclined turbulent jet represents the coolant flow and the cross-flow represents the hot combustion gases. In this configuration, it is known that the coolant jet tends to lift off the wall that is to be cooled, thus decreasing heat transfer effectiveness. The micro-ramp vortex generator is placed downstream of the coolant jet and is used to modify the trajectory of the coolant jet such that it remains closer to the wall, thus enhancing heat transfer. The purpose of this study is to examine the micro-ramp's effect on both the flowfield and heat transfer of the film cooling process. The coolant jet is inclined at an angle of 35 degrees to the freestream, the blowing ratio is 1.5, and the Reynolds number based on the jet diameter and freestream cross-flow velocity is 8000. The incompressible Navier-Stokes equations are solved numerically using a 3D finite volume solver (CU-FLOW) implemented on a Graphics Processing Unit (GPU).

1Supported by HEL-JTO through AFOSR Grant FA 9550-07-1-0504

3:10PM RD.00006 Generalized Velocity Boundary Conditions for Kleiser and Schumann's Influence-Matrix Method , XIAOFENG LIU, Dept. of Civil and Environmental Engineering, University of Texas at San Antonio — Kleiser and Schumann (1980) introduced a novel influence-matrix method to treat the incompressibility and no-slip boundary conditions when solving the Navier-Stokes equations. They also outlined the related "tau" error correction technique which is essential for the high accuracy direct numerical simulation (DNS) of turbulent flows. Werne (1995) proposed a revised "tau" correction algorithm on the "A"-problem level. Both methods are correct, though some technical differences exist. Note also that both methods are specific for the no-slip boundary conditions where the odd and even modes of Chebyshev expansion in the wall normal direction are decoupled. In this talk, the Kleiser and Schumann method will be generalized to treat the Robin type velocity boundary conditions and the related "tau" error corrections. This new method will broaden the applicability of the Kleiser and Schumann method to situations where velocity boundary conditions are not limited to no-slip. Three examples (channel flow with a free surface, density current in an open channel, drag reduction in a hydrodynamic channel) will be shown with extensive validations using various statistics of turbulent flow. All examples show excellent agreement with data in the literature and the velocity field is divergence free up to machine precision.

4:23PM RD.00007 Microscopic analysis and simulation of check-mark stain on the galvanized steel strip , HONGYUN SO, HYUN GI YOON, MYUNG KYOON CHUNG, Korea Advanced Institute of Science and Technology (KAIST) — When galvanized steel strip is produced through a continuous hot-dip galvanizing process, the thickness of adhered zinc film is controlled by plane impinging air jet referred to as "air-kine system". In such a gas-jet wiping process, stain of check-mark or sag line shape frequently appears. The check-mark defect is caused by non-uniform zinc coating and the oblique patterns such as "V", "V" or "X" on the coated surface. The present paper presents a cause and analysis of the check-mark formation and a numerical simulation of sag lines by using the numerical data produced by Large Eddy Simulation (LES) of the three-dimensional compressible turbulent flow field around the air-kine system. It was found that there is alternating plane-wise vortices near the stagnation line region and such alternating vortices move almost periodically to the right and to the left sides on the stagnation line due to the jet flow instability. Meanwhile, in order to simulate the check-mark formation, a novel perturbation model has been developed to predict the variation of coating thickness along the transverse direction. Finally, the three-dimensional zinc coating surface was obtained by the present perturbation model. It was found that the sag line formation is determined by the combination of the instantaneous coating thickness distribution along the transverse direction near the stagnation line and the feed speed of the steel strip.

1This work was funded in part by NASA GSRP Fellowship under grant number NNX08AY63H.

This work was supported by a NASA GSRP Fellowship under grant number NNX08AY63H.
that contains two species of interacting, elastically-active micellar chains. 

Despite more than two decades of HIV vaccine research, there is still no efficacious HIV vaccine. Very recently, a research group has shown that a microbicide gel formulation of antiretroviral drug Tenofovir, significantly inhibits HIV transmission to women [1]. However, there is a widespread agreement that more effective and diverse drug delivery vehicles must be developed. In this setting, there is now great interest in developing different delivery vehicles such as vaginal rings, gels, and films. Here, we develop a model for transient fluid uptake and swelling behavior, and subsequent dissolution and drug deployment from a film containing anti-HIV micробicide. In the model, the polymer structural relaxation via water uptake is assumed to follow first order kinetics. In the case of a film loaded with an osmotically active solute, the kinetic equation is modified to account for the osmotic effect. The transport rate of solvent and solute within the matrix is characterized by a diffusion equation. After the matrix is relaxed to a specified concentration of solvent, lubrication theory and convective-diffusive transport are employed for flow of the liquefied matrix and drug dispersion respectively. [1] Karim, et al., Science, 2010.

1 NIH U19 AI 077289

3:18PM RF.00002 Coating flow of an anti-HIV microbicid gel: boundary dilution and yield stress

1, ANDREW J. SZERI, SAVAS TASOGLU, SU CHAN PARK, UC, Berkeley, DAVID F. KATZ, Duke University — A recent study has confirmed, for the first time, that a vaginal gel formulation of the antiretroviral drug Tenofovir, when topically applied, significantly inhibits sexual HIV transmission to women [1]. However, the gel for this drug, and anti-HIV microbicid gels in general, have not been designed using an understanding of how gel spreading govern successful drug delivery. Elastohydrodynamic lubrication theory can be applied to model spreading of microbicid gels [2]. Here, we extend our initial analysis: we incorporate a yield stress, and we model the effects of gel dilution due to contact with vaginal fluid produced at the gel-tissue interface. Our model developed in [2] is supplemented with a convective-diffusive transport equation to characterize dilution, and solved using a multi-step scheme in a moving domain. The association between local dilution of gel and rheological properties is obtained experimentally. To model the common yield stress property of gels, we proceed by scaling analysis first. This establishes the conditions for validity of lubrication theory of a shear thinning yield stress fluid. This involves further development of the model in [2], incorporating a biviscosity model.


1 NIH U19 AI 077289

3:31PM RF.00003 Motion of Non-Newtonian liquid plugs in channels

1, PARSA ZAMANKHAN, University of Michigan - Department of Biomedical-Engineering, BRIAN HELENBROOK, Department of Mechanical and Aeronautical Engineering, Clarkson University, SUICHI TAKAYAMA, JAMES GROTBERG, University of Michigan - Department of Biomedical-Engineering — Some major transport phenomena in the human respiratory system such as the reopening of the occlude airways and drug delivery involve with propagation of liquid plugs, constituted from non-Newtonian fluids. In this presentation the transport of liquid plugs, constituted from the yield stress Bingham, and shear thinning power-law fluids is investigated numerically. The governing equations are solved by a mixed-discontinuous finite element formulation while the free surface is resolved by the method of spines. The constitutive equation for Bingham fluid is implemented through a regularization method. The effects of the yield stress and the power-law index on the flow feature are compared and discussed. Special attention is given to the distribution of the stresses along the wall with applications in cell injury studies.

1Supported by NIH 84370 and grant HL65156.

3:44PM RF.00004 ABSTRACT WITHDRAWN —

3:57PM RF.00005 Non-Newtonian Fluids Spreading with Surface Tension Effect: 3D Numerical Analysis Using FEM and Experimental Study

BIN HU, SARAH KIEWEG, University of Kansas — Gravity-driven thin film flow down an incline is studied for optimal design of polymeric drug delivery vehicles, such as anti-HIV topical microbicidcs. We develop a 3D FEM model using non-Newtonian mechanics to model the flow of gels in response to gravity, surface tension and shear-thinning. Constant volume setup is applied within the lubrication approximation scope. The lengthwise profiles of the 3D model agree with our previous 2D finite difference model, while the transverse contact line patterns of the 3D model are compared to the experiments. With incorporation of surface tension, capillary ridges are observed at the leading front in both 2D and 3D models. Previously published studies show that capillary ridge can amplify the fingering instabilities in transverse direction. Sensitivity studies (2D & 3D) and experiments are carried out to describe the influence of surface tension and shear-thinning on capillary ridge and fingering instabilities.

4:10PM RF.00006 Flow of wormlike micelles in confined geometry

NICOLAS LOUVET, CHLOÉ MASSELON, ANNIE COLIN, Rhodia - Laboratory of the Future — We study wormlike micelles flowing in confined geometry to study the local rheology of such fluids. Experiments show that the properties of such fluids undergoing a strong shear stress gradient can only be described by an equation including non-local terms. Then the flow of very long wormlike micelles is studied both in microfluidic channel and in Couette geometry coupling with ultrasonic velocimetry.

4:23PM RF.00007 Modeling of irreversible flow-induced gelation in wormlike micellar solutions

NEVILLE DUBASH, JOSHUA CARDIEL, PERRY CHELING, AMY SHEN, University of Washington — Wormlike micellar solutions are known to exhibit a variety of interesting phenomena, one of which is the formation of gel-like structures under simple flow conditions. Previously, these flow-induced structures were all observed to be temporary, and the gels would dissipate upon cessation of the flow. Recently, however, it has been shown that it is possible to produce a purely flow-induced irreversible transformation in certain micellar solutions. This irreversible gel formation is brought about via a mixed shearing/extensive flow in which the fluid experiences very high rates of strain and total strains. Here, we examine this gelation phenomenon and present a model which is able to capture the irreversible rheological changes observed in our experimental system. The model is based on an existing network scission model for micellar solutions that contains two species of interacting, elastically-active micellar chains.

the adjacent larger scales. In the third case of equally strong rotation and stratification, there are only slightly anisotropic constant fluxes of energy and potential enstrophy, mostly to small-scales in three different regimes of stratification and rotation. For strongly stratified flow with moderate rotation, we observe constant fluxes of both flows, forced in the large-scales, with fixed rotation and stable stratification along the vertical axis, to study the cascades of energy and potential enstrophy.

4:36PM RG.00008 Switching and defect dynamics in liquid crystal devices. ADRIANO TIRIBOCCHI, GIUSEPPE GONNELLA, University of Bari, Department of Physics and INFN - Italy, DAVIDE MARENDUZZO, SUPA, School of Physics, University of Edinburgh - Scotland, ENZO ORLANDINI, University of Padova, Department of Physics and INFN - Italy — We present some numerical results about nematic cells in which an external electric field is applied. We show that it is possible to design a simple two-domain hybridly aligned nematic cell which is bistable and we elucidate the role of hydrodynamics by using a lattice Boltzmann approach. Moreover we report some results of electric field induced switching of devices built starting from cholesteric blue phase, showing how various disclination patterns can be predicted. Hydrodynamic effects are also observed to affect the switching dynamics.

Tuesday, November 23, 2010 3:05PM - 4:49PM — Session RG Stratified Flows II — Long Beach Convention Center 103B

3:05PM RG.00001 Steady solutions for plumes in non-uniform stratifications. NIGEL KAYE, Clemson University, MATTHEW SCASE, University of Nottingham — The plume conservation equations of Morton et al. (1956) are recast in terms of the plume radius, flux balance parameter Ω, and a dimensionless parameter that characterizes the stratification. This set of equations lead to simple analytic solutions for steady straight-sided plumes in non-uniformly stratified environments. Steady plumes in non-uniform stratification can occur for both stable (Caulfield & Woods 1998) and unstable (Batchelor 1954) stratifications whose strength has a power law variation with height. We present analytic solutions for the range of stratification power-law decay rates κ for which straight sided plumes are possible. The approach used provides significant physical insight into the limits on κ that permit straight-sided solutions. We also present analytic solutions for the power law behaviour with height of the fluxes of volume, momentum and buoyancy. This result demonstrates that the models of Batchelor and Caulfield & Woods are two halves of the same continuum of solutions. The flux power law behavior explains the transition between the Batchelor solutions and the Caulfield & Woods solutions that occurs when κ = −8/3. For κ < −8/3 the buoyancy flux decays with height and, therefore, the stratification must be stable. Whereas for κ > −8/3 the buoyancy flux must increase with height, requiring an unstable stratification.

3:18PM RG.00002 3D Vortices in Protoplanetary Disks. SAMY KAMAL, JOSEPH BARRANCO, San Francisco State University, PHILIP MARCUS, University of California, Berkeley — Like the atmosphere of Jupiter, protoplanetary disks (thin disks of gas & dust in orbit around newly-formed stars) are characterized by rapid rotation and intense shear, inspiring proposals that disks may also be populated with long-lived, robust storms analogous to the Great Red Spot. Such vortices may play key roles in the formation of stars and planets by transporting angular momentum, as well as trapping and concentrating dust grains, feeding the formation of planetesimals, the “building blocks” of planets. In our previous work (Barranco & Marcus 2005), we showed via numerical simulation (with an anelastic spectral code) that vortices near the midplane of the disk suffer an antisymmetric instability and are destroyed. However, internal gravity waves propagate away from the midplane, amplify and break, creating bands of vorticity that roll-up into new long-lived, stable vortices above and below the midplane. We will present new results on 3D vortex dynamics in protoplanetary disks, exploring the role of factors unique to this context: the Coriolis parameter f, the shear rate σ, and the Brunt-Väisälä frequency N are all of the same order of magnitude. In the region around the midplane N < f, whereas a few pressure scale heights off the midplane, there is a transition to N > f. This leads to strong refraction of internal gravity waves, causing the waves to amplify and break, generating vorticity.

3:31PM RG.00003 Stratified Flows with Vertical Layering of Density: Theoretical and Experimental Study of the Time Evolution of Flow Configurations and their Stability. MATTHEW MOORE, Courant Institute of Mathematical Sciences, ROBERTO CAMASSA, DAVID HENDEL, RICHARD M. MCLAUGHLIN, MARSHALL NEWMAN, University of North Carolina Chapel Hill, KUAI YU, North Carolina State University, UNC RTG FLUIDS GROUP TEAM — A vertically moving boundary in a stratified fluid can create and maintain a horizontal density gradient or vertical layering of density. Such a flow is created experimentally by towing a narrow fiber upwards through an initially stable stratification, as a layer of heavier fluid entrained by the fiber forms a vertical column. We develop a lubrication model to predict the time evolution which shows close agreement with the experiment. We perform stability analysis on a class of vertically layered shear flows and find a critical length-scale for the size of the entrained layer, below which the flow is stable and above which the flow is unstable. The bifurcation behavior is independent of the Reynolds number. Flows with unstable layer sizes have been created experimentally, however the small amplification rates prevent the instabilities from being observed.

1NSF RTG grant: DMS-0502266

3:44PM RG.00004 The Collapse of an Axisymmetric Mixed Patch and Internal Wave Generation in Uniformly Stratified Fluid. AMBER HOLDSWORTH, BRUCE SUTHERLAND, University of Alberta — Hurricanes are responsible for mixing localized patches of the upper ocean leaving cooler waters in their wakes. The region collapses into a stratified ambient forming an gravity current and generating internal waves beneath the mixed patch. In an effort to understand the axisymmetric collapse of a mixed patch into uniformly stratified fluid laboratory experiments are performed and wave properties are determined using a non-intrusive technique called Synthetic Schlieren. We find internal wave frequencies are set by the buoyancy frequency, (ω ≈ 0.8Nz) and that the horizontal wavelength is set by the radius of the cylinder so that kh ≈ 2Hz. Vertical displacement amplitudes scale with the depth of the mixed patch according to |z|/Hm = 0.016 ± 0.001 and we find that about 2% of the available potential energy of the mixed region is extracted by vertically propagating internal waves. The work presented here is a precursor to the more complicated rotating case which will more realistically simulate the oceanic case. Extrapolation of these results is certainly premature, but a conservative estimate of the energy extracted by internal waves through the process of mixed region collapse is on the order of 1 GW. That is an estimated 2 TW of power over the generation time and is comparable to the power exerted by tides and winds over the ocean.

3:57PM RG.00005 Joint downscale fluxes of energy and potential enstrophy in rotating stratified Boussinesq flows. SUSAN KURIEN, HUSSEIN ALUIE, Los Alamos National Laboratory — We use high-resolution simulations of Boussinesq flows, forced in the large-scales, with fixed rotation and stable stratification along the vertical axis, to study the cascades of energy and potential enstrophy to small-scales in three different regimes of stratification and rotation. For strongly stratified flow with moderate rotation, we observe constant fluxes of both energy and potential enstrophy into fourier modes with large vertical component kz, while being entirely suppressed in modes with large horizontal component kx. The fluxes in this regime are predominantly due to a highly non-local transfer from the large-scales directly to the smallest scales. On the other hand, for strongly rotating flow with moderate stratification, there are constant fluxes of energy and potential enstrophy to modes with large kx while being completely suppressed to modes with large kz. We find that the fluxes in this regime are due to a “diffusely” local transfer much like in isotropic Navier-Stokes turbulence. Extrapolation of these results is certainly premature, but a conservative estimate of the energy extracted by internal waves through the process of mixed region collapse is on the order of 1 GW. That is an estimated 2 TW of power over the generation time and is comparable to the power exerted by tides and winds over the ocean.
4:10PM RG.00006 Stability of a pancake vortex in a stratified fluid. M. ELETTA NEGRETTI, PAUL BILLANT, LadHyX, CNRS, Ecole Polytechnique, France — Vortices in stably stratified fluids have generally a pancake shape with a small vertical thickness compared to their horizontal size. Such vortices exhibit a high vertical shear which may induce Kelvin-Helmholtz instabilities. The pressure and density anomaly in their core might trigger also gravitational instabilities. In order to understand which mechanism determines the minimum thickness of the vortex, we investigate the three-dimensional linear stability of an axisymmetric pancake vortex in a stably stratified fluid. The angular velocity of the base flow has a Lamb-Oseen radial profile with a Gaussian distribution in the vertical direction. We find that the vortex becomes unstable when the aspect ratio is below a critical value, which scales with the Fröuide number. We show that the instability is gravitational by looking at the classical criteria for each instability, which predict larger critical aspect ratios for the gravitational instability as compared to the shear instability. The numerical results agree well with the gravitational instability theory. We have generalized this result to any vertical distribution of the angular velocity and almost any profile of the vortex. We show that the properties of the gravitational instability can be explained by considering an unstably stratified fluid in solid body rotation. The influence of the Reynolds number will be also discussed.

4:23PM RG.00007 Vortex dynamics in a wave field. GAELE PERRET, ADRIEN POUJARDIN, JEROME BROSSARD, LOMC - CNRS FRE3102, Université du Havre — The interaction of waves and current with submerged structures in coastal zones generates some complex hydrodynamics features which may considerably impact the local environment. The geometrical singularities of the structures produce concentrated vortex filaments which may impact the sea bed and/or the free surface. The objective of the present study is to characterize the vortex dynamics generated by a horizontal plate considered as a vortex generator, in a regular wave field. Vortices are generated at the edges of the plate. They undergo three-dimensional instabilities leading to their destruction thanks to laboratory experiments conducted in two different wave flumes to study the impact of the scale on the dynamics. The two-dimensional vortex dynamics is characterized using PIV measurements. Vortex intensity, trajectory and life time are determined. The three-dimensional dynamics is studied thanks to stereo photography. The vortices are visualised with hydrogen bubbles generated at the edges of the plate by electrolyse. The evolution of the vortices is visualized by two CCD cameras located in different planes. Two most unstable wavelengths are observed which do not seem to depend on the width of the wave flume.

4:36PM RG.00008 Numerical simulation of flow around a sphere moving through a stratified fluid. TREvor ORR, JULIAN D0MARAÐZKI, University of Southern California, GEORGE CONSTANTINESCU, University of Iowa — Flows generated by submerged bodies in stratified fluids have been investigated in numerous experiments and numerical simulations. Numerical simulations are focused in the far-wake region, where computational costs prohibit including the explicit computation of the flow around the sphere. Initial conditions for such far-wake simulations are constructed using information gathered from experimental results of the near-wake properties, but these initializations lack full information about the density field. We present results of numerical simulations that explicitly include the sphere in the computational domain over a range of Reynolds numbers and Fröuide numbers. The simulations are compared with existing experimental and numerical data. In particular, turbulent simulations using the Spalt-Allmaras method are included along with comparison of experimental data collected at USC for Re=9000 and Fr=4.

Tuesday, November 23, 2010 3:05PM - 4:10PM – Session RH Convection and Buoyancy Driven Flows VI Long Beach Convention Center 103C

3:05PM RH.00001 Hots spots in fingering of exothermic autocatalytic chemical fronts. G. GERARD, P. GROSFILS, A. DE WIT, Université Libre de Bruxelles, Belgium, T. TÖTH, D. HORVÁTH, A. TÖTH, Szeged University, Hungary — Across traveling autocatalytic fronts, density differences can result from composition and temperature changes. These density differences lead to buoyancy-driven hydrodynamic instabilities when the heavier solution overlies the lighter one. Using combined experimental and theoretical approaches, we examine the properties of the temperature field around such a buoyantly unstable exothermic autocatalytic front in presence of heat losses. Experimentally, the dynamics of the chlorite-tetrathionate reaction are studied in a Hele-Shaw cell. The concentration field is observed by a color indicator while the two-dimensional thermal field is obtained by an interferometric technique. Because of the heat losses, products are cooled down behind the reaction zone. The interferometric analysis moreover reveals the presence of hot spots, i.e., local areas where the temperature is larger than in a stable front. To understand the properties of the hot spots, we have developed a theoretical model coupling the evolution of the concentration and temperature fields to that of the velocity field. We show that hot spots exist in the presence of buoyancy-induced convection only if heat diffuses faster than mass and/or in presence of heat losses. We quantify the maximum value of temperature obtained in presence of convection as a function of the various parameters of the problem.

3:18PM RH.00002 Hydrodynamic Instabilities of Acid-Base Reaction Fronts: Active Role of a Color Indicator. L.A. RIOLO, C. ALMARCHA, P.M.J. TREVELYAN, NLPC, Universite Libre de Bruxelles, Belgium, C. EL HASI, A. ZALTS, UNGS, Argentina, A. D’ONOFRIO, GMP, Universidade de Buenos Aires, Argentina, A. DE WIT, NLPC, Universite Libre de Bruxelles, Belgium — Chemical reactions are able to trigger hydrodynamic flows by, for example changing the density of the solutions across the reactive interfaces. In this work we present an experimental and theoretical study of the buoyancy-driven hydrodynamic instabilities that can occur when two miscible reactive solutions of an acid-base system are put in contact in the gravity field. We compare situations where a hydrochloric acid aqueous solution is put on top of a sodium hydroxide aqueous solution with or without a color indicator (Bromocresol Green). We also analyze the situation where a hydrochloric acid is put on top of an aqueous solution of a color indicator without any base. We show that the patterns observed and the instabilities taking place strongly depend on the presence of a color indicator. Using a reaction-diffusion model for the concentrations of all species (including the color indicator) we analyze the different possible sources of destabilization of the acid-base front and explain the various instabilities observed in each experimental system.

3:31PM RH.00003 Numerical study of thermo-solutal convection induced by evaporation. BENOÎT TROUETTE, ERIC CHÉNIER, CLAUDINE DANG VU DELCARTE, FRÉDÉRIC DOUMENC, BÉATRICE GUERRIER — During the drying of a volatile solution, both thermal and mass effects are coupled. The instability mechanism is first of the solutal Marangoni type. In order to understand why experimentalists observe thermal convective patterns at the beginning of the process, numerical studies are performed and thermal and mass effects are separately studied. In both cases, convection is considered as significant when the Péclet number ($Pe$) is greater than 1. The time evolution of $Pe$ is studied to explore the transient character of the problem. A stability map as a function of various physical mechanisms that affect evaporating liquid films. The present work extends the evolution equation proposed in literature by including the effect of buoyancy through the Boussinesq approximation. The extended evolution equation allows for capturing buoyancy effects inherent in evaporating thick liquid films.

3:44PM RH.00004 Inclusion of buoyancy effects in the evaporating liquid film evolution equation. ANEET NARENDRANATH, Michigan Technological University, JERAMY KIMBALL, JAMES HERMANSON, University of Washington, ROBERT KOLKKA, JEFFREY ALLEN, Michigan Technological University — Macroscopic liquid films are entities that are important in biophysics, physics, and engineering, as well as in nature. They can be composed of common liquids such as water or oil, rheologically complex materials such as polymers solutions or melts, or complex mixtures of phases or components. When the films are subjected to the action of various mechanical, thermal, or structural factors, they undergo three-dimensional instabilities leading to their destruction thanks to laboratory experiments conducted in two different wave flumes to study the impact of the scale on the dynamics. The two-dimensional vortex dynamics is characterized using PIV measurements. Vortex intensity, trajectory and life time are determined. The three-dimensional dynamics is studied thanks to stereo photography. The vortices are visualised with hydrogen bubbles generated at the edges of the plate by electrolyse. The evolution of the vortices is visualized by two CCD cameras located in different planes. Two most unstable wavelengths are observed which do not seem to depend on the width of the wave flume.
3:57PM RH.00005 Hydrothermal waves in evaporating annular pools and sessile drops using DNS. PEDRO SAENZ, PRASHANT VALLURI, University of Edinburgh, GEORGE KARAPETAS, Imperial College London, KHELLIL SEFIANE, University of Edinburgh, OMAR MATAR, Imperial College London — Thermocapillary effects generated due to thermal gradients in annular liquid pools and resulting in hydrothermal waves under inert, saturated and evaporating atmospheres are investigated using two-phase direct numerical simulations in 3D. For annular pools under inert environments, the volume-of-fluid method is used to capture the interface, with special attention towards the grid resolution near the vapour-liquid interface. The results show that the interface temperature distribution follows a regular azimuthal pattern, representative of hydrothermal wave structures, along with small-amplitude interfacial waves. The effects of evaporation fluxes and the interfacial depths on the linear (early-time) and non-linear (late-time) development of hydrothermal temperature and interfacial waves will be presented. Under inert environments, the azimuthal structures qualitatively agree with experimental and numerical studies (with a single-phase model and a non-deformable free surface) of Schwabe et. al. (2003). Evaporating sessile droplets simulated using diffuse-interface method will be presented and compared against analytical integral balance models.

1Supported by the NRL and WCU Programs, KRF, MEST, Korea.

Tuesday, November 23, 2010 3:05PM - 4:23PM —
Session RJ Flow Control VII Long Beach Convention Center 201A

3:05PM RJ.00001 Detection and estimation of the instantaneous flow topology on an airfoil using proper orthogonal decomposition. JURGEN SEIDEL, CASEY FAGLEY, TOM MCLAUGHLIN, US Air Force Academy — For the control of the lift distribution on a wing, the local flow state has to be known instantaneously, in particular, the location of critical points in the flow topology such as stagnation point, separation point and reattachment point. Unsteady CFD simulations are used to determine the flow field around a Naca 0018 airfoil at moderate Reynolds number. These simulations are then analyzed using Proper Orthogonal Decomposition (POD) to develop a database of flow states at a wide range of angles of attack. In addition, POD is performed using data on the airfoil surface. A mapping between the two databases is used to develop a global flow state estimator. Robustly estimating the critical points in the flow topology in real time allows for the formulation of a Reduced Order Model (ROM) which relates flow field characteristics and surface data, including the effect of controlled forcing input. The accuracy of this model and its efficacy for developing feedback control strategies for the control of the lift distribution are determined.

3:15PM RJ.00002 Linear proportional-integral control of turbulent channel flow for drag reduction. EUIYOUNG KIM, HAECHEON CHOI, Seoul National University — Choi, Moin & Kim (1994, JFM) applied an opposition control, \( v_{\text{up}} = -v_{y+10} \), to turbulent channel flow and obtained about 25 % drag reduction, where \( v_{\text{up}} \) is the blowing and suction at the wall, and \( y \) is the wall-normal velocity. The idea in that study was to provide a distributed blowing/suction at the wall opposite to the induced motion by the near-wall streamwise vortices and to reduce their strength, resulting in drag reduction. In the present study, we reconsider this control problem from the view point of linear proportional-integral-differential control. The opposition control by Choi et al. (1994) is a proportional control and thus contains steady-state errors. In other words, the target sensing velocity does not go to zero (\( v_{y+10} \neq 0 \)) even after control. To reduce this steady-state errors, we introduce a proportional-integral (PI) control, \( v_{\text{up}} = -\alpha v_{y+} - \beta \int v_{y+} \, dt \), where \( \alpha \) and \( \beta \) are the feedback gains, and \( y_+ \) is the sensing location above the wall. As a result of applying the PI control, the steady-state errors are significantly reduced and the effective sensing region becomes wide. The detailed results by varying the feedback gains and sensing location will be shown in the presentation.

3:31PM RJ.00003 Adaptive Observation with Vehicle Dynamics. DAVID ZHANG, THOMAS BEWLEY, UCSD — Adaptive Observation (AO) studies mobile sensor deployment strategies to improve the estimation and forecast of various physical systems. Of the many approaches to the AO problem, few incorporate the dynamics of moving sensors into the trajectory planning algorithm. We propose a new AO algorithm which plans trajectories such that vehicle dynamics are inherently satisfied.

3:44PM RJ.00004 Experimental study on a three-dimensional riblet with particle image velocimetry. HIDEYUKI MIKI, KAORU IWAMOTO, AKIRA MURATA, Tokyo University of Agriculture and Technology — Experimental study on a new three-dimensional (3-D) blade riblet is carried out in a two-dimensional channel. The lateral spacing of our 3-D riblet surface is periodically changed in the streamwise direction. The flow structure over the 3-D riblet was analyzed in the turbulent flow field by using 2-D Particle Image Velocimetry (PIV) on a vertical (\( x - y \)) and a horizontal (\( x - z \)) plane. The turbulence statistics were compared with the corresponding flow over the flat surface in an attempt to identify the drag-reduction mechanism. Under a drag-reducing condition, the mean velocity profile showed upward shift in the log-law region. The streamwise, spanwise velocity fluctuations and the Reynolds shear stress were decreased, whereas the wall-normal velocity fluctuation was increased. The quadrant analysis of the Reynolds shear stress provides detailed information on the contributions to the total turbulence production from various events occurring in the flows. The 3-D riblets intensified the Reynolds shear stress producing event (second and forth quadrants). On the other hand, it was interesting to note that the first (outward) and third (inward) quadrants are dramatically increased compared with the smooth surface, leading to the drag-reducing effect.

3:57PM RJ.00005 Real-time turbulent plume estimation with mobile sensors. THOMAS BEWLEY, CHRISTOPHER COLBURN, DAVID ZHANG, UC San Diego, JOSEPH CESSNA, SAIC, NICHOLAS MOROZOVSKY, ANDREW CAVENDER, CHRISTOPHER SCHMIDT-WETEKAM, UC San Diego — Ensemble methods for estimating turbulent fluid systems are efficient methods for quantifying uncertainties in nonlinear, high-dimensional systems. Many real-time estimation algorithms for large-scale fluid systems have been tested and validated by the weather/oceanic forecasting communities, but (generally speaking) these methods have not been used for short time-scale and short length-scale models. We present estimation results for a contaminant plume release experiment. In this experiment, a passive scaler is released at a known location in a small domain and a turbulent environment. Mobile robots are deployed to measure wind velocity and plume concentration. Measurements are assimilated to estimate the wind field and quantify the uncertainty in the estimate, which is then used to plan waypoints for future measurements.

4:10PM RJ.00006 Game-theoretic Kalman Filter. CHRISTOPHER COLBURN, THOMAS BEWLEY, UC San Diego — The Kalman Filter (KF) is celebrated as the optimal estimator for systems with linear dynamics and gaussian uncertainty. Although most systems of interest do not have linear dynamics and are not forced by gaussian noise, the KF is used ubiquitously within industry. Thus, we present a novel estimation algorithm, the Game-theoretic Kalman Filter (GKF), which intelligently hedges between competing sequential filters and does not require the assumption of gaussian statistics to provide a “best” estimate.
The dynamics of a lipid vesicle in shear flow, HONG ZHAO, ERIC S.G. SHAQFEH, Stanford University — The dynamics of a lipid vesicle in a simple shear flow, where the lipid membrane is modeled as a two dimensional incompressible fluid with bending stiffness, is solved by a high-fidelity spectral boundary integral formulation. We combine our direct numerical simulation (DNS) with a linear stability analysis to solve the exact critical internal/external viscosity ratio for the transition from the steady tank-treading motion to the unsteady tumbling/tumbling motions at different shear rates. It is demonstrated that a fourth (and higher) order spherical harmonic expansion of the vesicle shape is necessary for obtaining quantitatively correct transition boundaries. The particle stresslets in different flow regimes are calculated, and the consequences for the rheology of a dilute suspension is discussed. In addition, our DNS reveals a family of time-periodic and out-of-plane vesicle motion patterns, where the orientation of principle axes follow orbits that resemble but are fundamentally different from the classical Jeffery orbits of rigid particles due to the vesicle's deformability. The effect of wall boundaries on the vesicle motion is then investigated within our DNS by using the known Green’s function for a no-slip walls at zero Reynolds number. It is demonstrated that wall interactions have a strong effect on the dynamic phase boundaries and stresslet. We finish by discussing the effect of thermal fluctuations and strategies of performing Brownian dynamics for the vesicle system.

Dynamics of a compound vesicle: numerical simulations, YUAN-NAN YOUNG, New Jersey Institute of Technology, SHRavan VEERAPANI, New York University, JERZY BLAWZDZIEwICZ, Texas Tech University, PETIA VLAHOVSka, Brown University — Extensive work, both numerical simulations and analytical modeling, on these dynamics provide insights to understanding the suspension phenomena of vesicles in experiments. Recently, they have been used as a multi-functional platform for drug-delivery. In this work the dynamics of such compound vesicles is investigated analytically using the small-deformation method. Results show that for a vesicle enclosing a rigid particle in a simple shear flow, transition from tank-treading to tumbling is possible even in the absence of viscosity mismatch in the interior and exterior fluids. Comparison with results from numerical simulations will be presented, and the rheology of suspension of such compound vesicles will be discussed.

Lattice-Boltzmann simulation of a confined tank-treading vesicle under shear, BADR KAOUI, Technische Universität Eindhoven (Eindhoven, The Netherlands) and CNRS - Université de Grenoble I (Grenoble, France), JENS HARTING, Technische Universität Eindhoven (Eindhoven, The Netherlands) and Universität Stuttgart (Stuttgart, Germany), CHAOUIJ MBSAH, CNRS - Université de Grenoble I (Grenoble, France) — Dynamics of a vesicle under shear flow between two parallel plates is studied using lattice-Boltzmann simulations. We first present how we adapted the lattice-Boltzmann method to simulate vesicle dynamics basing on the same approach as the one used in the immersed boundary method. The fluid flow is computed on an Eulerian regular fixed mesh while the location of the vesicle membrane is tracked by a Lagrangian moving mesh. As benchmarking tests, the known vesicle equilibrium shapes in a fluid at rest are found to be in excellent agreement. In the dynamical behavior of a vesicle under simple shear flow, we focus on investigating the effect of confinement on the dynamics. In particular we study how the vesicle’s steady inclination angle in the tank-treading regime depends on the degree of confinement (the ratio of the effective radius of the vesicle to the half height of the channel). The effective viscosity of the fluid, in the presence of the vesicle, is also measured and the influence of the confinement on it is analysed. Both the inclination angle and the membrane tank-treading velocity are found to decrease with increasing confinement.

Gravity Induced Sedimentation of Giant Lipid Vesicles, ANDRES GONZALEZ-MANCERA, IVAN REY SUAREZ, CHAD LEIDY, Universidad de los Andes — The mechanical properties of the lipid bilayer influence the gravity-induced sedimentation of vesicles toward a horizontal surface. In this work, the sedimentation rate and strain of lipid vesicles is studied using computational simulations performed using an algorithm based on the boundary element method. The mechanical behavior of the lipid bilayer is modeled considering two modes of deformation responsible for increases in area strain. The first is the smoothening of suboptical thermal undulations and the second is the direct stretching of the area per lipid molecule. Properties of the lipid bilayer are controlled by adjusting its bending and area compressibility moduli. The electrostatic interaction between the sedimenting vesicle and the glass surface is also considered in order to improve agreement with our experimental measurements. We use the linear Deryagin approximation, which takes into account ionic screening, to calculate the electrostatic repulsive interaction between the glass surface and the charged vesicle. The algorithm shows good agreement with experimental results for both the sedimentation rate and vesicle deformation at equilibrium.
3:46PM RK.00008 The effect of the electrical double layer on the membrane charging process, MIAO YU, HAO LIN, Rutgers University — The electrical charging process of a liposome membrane immersed in electrolytic solutions is of significance to a variety of applications including electroporation and electrodeformation. In these phenomena, the build-up of a potential difference across the membrane (the so-called transmembrane potential, or TMP) induces pore formation and membrane permeabilization (in electroporation) or deformation (in electrodeformation). The classical model treats the membrane as an equivalent capacitor-resistor system which is valid in the zero-thickness electrical double layer (EDL) limit. In this work, the effects of a finite EDL on the charging dynamics are investigated. Starting from the Nernst-Planck equations governing ionic transport, the membrane charging process is solved in both planar and spherical geometries, and using both analytical and numerical methods. The results demonstrate that the effects of the EDL become more significant as the electrical conductivity of the electrolytic solution decreases, which is a natural consequence of an increased Debye length. The steric effect, which often arises in the limit of large zeta-potentials, is shown to be insignificant for physiological applications. The effective circuit equivalence of the EDL is calculated and validated. The results are discussed in comparison with experimental data on electroporation from the literature.

Tuesday, November 23, 2010 3:05PM - 4:36PM — Session RL Biofluids: Physiological Cerebral — Long Beach Convention Center 202A

3:05PM RL.00001 Blending CFD simulations with clinical measurements1, VITALIY RAYZ, GABRIEL ACEVEDO-BOLTON, Radiology, UCSF, LOIC BOUSSEL, Creatis-LRMM (LB, PCD), Lyon, France, DAVID SALONER, Radiology, UCSF — Patient-specific CFD models accurately capture complex flows in aneurysmal arteries and predict flow-derived parameters affecting disease progression, such as wall shear stress (WSS). A disadvantage of CFD is convergence time. Blood flow can also be measured in vivo with phase-contrast magnetic resonance velocimetry (PC-MRV). This method of the pulsatile blood flow in a vascular model is a valuable tool for evaluating the accuracy of CFD models. PC-MRV with CFD in order to quickly achieve an accurate solution. PC-MRV data obtained for 3 cerebral aneurysm patients were used as initial and boundary conditions for CFD simulations carried out in the same geometries. Lower-resolution MR data were interpolated into a finer computational mesh. Simulation time was reduced in all cases and excellent agreement was observed between the flow fields obtained with this technique and those obtained with fully convergent simulations started from zero initial conditions. The proposed method can help clinicians obtain relevant quantitative data in just a few hours after imaging.

1NIH Grants K25NS058981, K25NS058573

3:18PM RL.00002 Cerebral aneurysms: relations between geometry, hemodynamics and aneurysm location in the cerebral vasculature, TIZIANO PASSERINI, ALESSANDRO VENEZIANI, Emory University, LAURA SANCHALI, PIERCESARE SECCHI, SIMONE VANTINI, Politecnico di Milano — In cerebral blood circulation, the interplay of arterial geometrical features and flow dynamics is thought to play a significant role in the development of aneurysms. In the framework of the Aneurisk project, patient-specific morphology reconstructions were conducted with the open-source software VMTK (www.vmtk.org) on a set of computational angiography images provided by Ospedale Niguarda (Milano, Italy). Computational fluid dynamics (CFD) simulations were performed with a software based on the library LifeV (www.lifev.org). The joint statistical analysis of geometries and simulations highlights the possible association of certain spatial patterns of radius, curvature and shear load along the Internal Carotid Artery (ICA) with the presence, position and previous event of rupture of an aneurysm in the entire cerebral vasculature. Moreover, some possible landmarks are identified to be monitored for the assessment of a Potential Rupture Risk Index.

3:31PM RL.00003 A Mathematical Model of Intracranial Saccular Aneurysms: Evidence of Hemodynamic Instability, MICHAEL CALVISI, University of Colorado at Colorado Springs, STEPHEN DAVIS, MICHAEL MIKSIK, Northwestern University — Intracranial saccular aneurysms tend to form at the apex of arterial bifurcations and often assume a nominally spherical shape. In certain cases, the aneurysm growth can become unstable and lead to rupture. While the mechanisms of instability are not well understood, hemodynamics almost certainly play an important role. In this talk, a mathematical model of a saccular aneurysm is presented that describes the shape deformations of an initially spherical membrane interacting with a viscous fluid in the interior. The governing equations are derived from the equations of a thin shell complemented with constitutive models that are representative of aneurysmal tissue. Among the key findings are that two families of free vibration modes exist and, for certain values of the membrane properties, one family of nonspherical, axisymmetric modes is unstable to small perturbations. In addition, the presence of a vortical interior flow of sufficient strength can excite resonance of the membrane – an unstable phenomenon that might cause eventual rupture.

3:44PM RL.00004 Volumetric Velocity Measurements of Pulsating Flow through a Model Aneurysm1, DANIEL TROOLIN, Fluid Mechanics Division, TSI Incorporated, DEVESH AMATYA, ELLEN LONGMIRE, Aerospace Engineering and Mechanics, University of Minnesota — Volumetric 3-component velocimetry (V3V) was used to examine the flow structure inside of a scaled-up transparent urethane model of a saccular aneurysm. The model was fabricated to match the geometry of an in vivo case. Index matching was used to minimize optical distortions caused by the curved walls of the model. The model and a surrounding visualization box were integrated into a custom-built pulse duplicator system with in-line flow meter and pressure transducers. The pulsing frequency and amplitude were controlled independently to generate two flow conditions each having a non-dimensional peak Reynolds (Re) and Womersley (Wo) Number: Re = 250, Wo = 10.4 and Re = 125, Wo = 7.4. Phase-locked and instantaneous measurements of the pulsatile flow upstream, downstream and within the aneurysm reveals accurate reconstruction of the flow dynamics. Plots and movies will be shown, and a detailed discussion of the flow and various experimental considerations will be included.

1Supported by NIH (R01NS042646-08).

3:57PM RL.00005 Validation of Blood Flow Simulations in Intracranial Aneurysms1, YUE YU, Brown University, TÖMER ANOR, Children's Hospital, HYOUNGSU BAEK, Brown University/MIT, MAHESH JAYARAMAN, Brown University Medical School, JOSEPH MADSEN, Children’s Hospital, GEORGE KARNIADAKIS, Brown University — Catheter-based digital subtraction angiography (DSA) is the most accurate diagnostic procedure for investigating vascular anomalies and cerebral blood flow. Here we describe utilization of DSA in a patient with an intracranial aneurysm to validate corresponding spectral element simulations. Subsequently, we examine via visualization the structure of flow in internal carotid arteries laden with three different types of aneurysms: (1) a wide-necked saccular aneurysm, (2) a narrower-necked saccular aneurysm, and (3) a case with two adjacent saccular aneurysms. We have found through high resolution simulations that in cases (1) and (3) in physiological conditions a hydrodynamic instability occurs during the decelerating systolic phase resulting in a high frequency oscillation (20-50 Hz). We use the in-silico dye visualization to discriminate among different physical mechanisms causing the instability and contrast their effect with case (2) for which an instability arises only at much higher flowrates.

1Supported by NSF OCI-0904288.
4:10PM RL.00006 Control volume based hydrocephalus research: analysis of human data. BENJAMIN COHEN, TIMOTHY WEI, RPI, ABRAM VOORHEES, Siemens Medical Imaging, JOSEPH MADSEN, TOMER ANOR, Harvard Medical School — Hydrocephalus is a neuropathophysiological disorder primarily diagnosed by increased cerebrospinal fluid volume and pressure within the brain. To date, utilization of clinical measurements have been limited to understanding of the relative amplitude and timing of flow, volume and pressure waveforms; qualitative approaches without a clear framework for meaningful quantitative comparison. Pressure volume models and electric circuit analogs enforce volume conservation principles in terms of pressure. Control volume analysis, through the integral mass and momentum conservation equations, ensures that pressure and volume are accounted for using first principles fluid physics. This approach is able to directly incorporate the diverse measurements obtained by clinicians into a simple, direct and robust mechanics based framework. Clinical data obtained for analysis are discussed along with data processing techniques used to extract terms in the conservation equation. Control volume analysis provides a non-invasive, physics-based approach to extracting pressure information from magnetic resonance velocity data that cannot be measured directly by pressure instrumentation.

4:23PM RL.00007 Effect of Parent Artery Geometry on Flow Through Cerebral Aneurysm. MORGAN NOWAK, Syracuse University, HIROSHI HIGUCHI, TOSHIKO NAKAYAMA, MAKOTO OHTA, Tohoku University, SYRACUSE UNIVERSITY COLLABORATION, TOHOKU UNIVERSITY COLLABORATION — The significance of parent artery geometry secondary flow is studied experimentally using PIV techniques. Several generalized models are used to obtain fundamental information on secondary flow structures, vortex dynamics and wall shear stress under a pulsatile flow. Several time-dependent flow field through a model with the aneurysm on the lateral side of a curved parent artery is compared with that through a straight parent artery. The different wall shear stresses and fundamental structures seen within the aneurysm confirm the importance of the parent artery configuration in the development of aneurysm secondary flow. The experimental results are analyzed and compared with the numerical simulations.

Tuesday, November 23, 2010 3:05PM - 4:49PM — Session RM Free Surface Flows: Interfacial Phenomena Long Beach Convention Center 202B

3:05PM RM.00001 Transition between a film flowing down a slope and a liquid sheet floating on a denser fluid: where is the grounding line? LAURENT LIMAT, YASAR ATAS, Laboratoire Matiere et Systemes Complexes, UMR 7057 of CNRS and Univ. Paris Diderot, Paris, France; OLIVIER DEVACHELLE1, Department of Earth, Atmospheric and Planetary Sciences, Massachusetts Institute of Technology, Cambridge, USA; JULIEN MOLUCHTAR, MATHIEU RECEVEUR, Laboratoire Matiere et Systemes Complexes, UMR 7057 of CNRS and Univ. Paris Diderot, Paris, France — We have investigated experimentally and with analytical calculations, the flow of a viscous liquid down a incline entering suddenly a liquid bath of higher density. This leads to the formation of a liquid sheet floating on the bath, the line of intersection of the substrate and liquid/liquid interface being called the grounding line in literature relative to ice field formation. At long enough time, and for small enough substrate inclination, the position of the grounding line and the angle of detachment of the flow are given by very simple expressions, that are in good agreement with our experiments. This angle of detachment depends only on the substrate inclination and on the relative density mismatch, while the distance between the grounding line and the initial shore position is proportional to the inverse of the inclination angle.

1on leave to Institut de Physique du Globe de Paris (IPGP)

3:18PM RM.00002 Formation of dynamic coherent structures by an ensemble of rigid particles, DENIS MELNIKOV, DAMITRI PUSHKIN, VALENTINA SHEVTSOVA, Microgravity Research Center, Free University of Brussels (ULB), Belgium — We report numerical studies of the surprising effect of formation of coherent particulate structures in the thermocapillary liquid bridge flow. The studied regimes of the flow are characterized by a hydrothermal wave travelling in the azimuthal direction. Although formation of these structures was discovered experimentally more than a decade ago, until now it has remained unexplained and was not reproduced numerically in physically realistic regimes. The particles are small (with Stokes number of the order of 10-6) and non-obtrusive. Usually such particles are expected to follow the flow. However, under certain conditions they create stable coherent structures. Those structures are dynamic and rotate azimuthally together with the travelling wave. The results reported are counterparts of our theoretical study of the physical mechanism leading to the formation of particulate coherent structures.

3:31PM RM.00003 Transition from Selective Withdrawal to Light Layer Entrainment in an Oil-Water System, JOEL HARTENBERGER, TIMOTHY O’HERN, STEPHEN WEBB, Sandia National Laboratories, DARRYL JAMES, Texas Tech University — Selective withdrawal refers to the selective removal of fluid of one density without entraining an adjacent fluid layer of a different density. Most prior literature has examined removal of the lower density fluid and the transition to entraining the higher density fluid. In the present experiments, a higher density liquid is removed through a tube that extends just below its interface with a lower density fluid. The critical depth for a given flow rate at which the liquid-liquid interface transitions to entrain the lighter fluid was measured. Experiments were performed for a range of different light layer silicone oils and heavy layer water or brine, covering a range of density and viscosity ratios. Applications include density-stratified reservoirs and brine removal from oil storage caverns. Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

3:44PM RM.00004 The effect of spreading on vertically directed jet impinging a sharp density interface. ALLEN BEAUNE1, PETER FRIEDMAN, University of Massachusetts Dartmouth — A large existing body of literature categorizes the flow behavior of negatively buoyant jets and fountains and characterizes their flow structure into distinct regimes and their maximum penetration depth predominately as a function of the Richardson number. In the present study, similar flow regimes have been identified and determined to be a function of Richardson number based on jet properties at the interface. This “interface Richardson number” increases as the jet is separated from the interface based on a jet spreading factor. The study uses immiscible fluids (silicone oil and a glycerin water mixture) with matched indices of refraction.

1This research is funded by Sandia National Laboratories, operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

2Graduate Research Assistant

3:57PM RM.00005 Drop orbiting in a circular hydraulic jump, LUC LEBON, ANTOINE FRULEUX, CLEMENT SAVARO, Laboratoire Matiere et Systemes Complexes, UMR 7057 of CNRS and Univ. Paris Diderot, Paris, France, CHRISTOPHE PIRAT, Laboratoire Physique Matiere Condensee et Nanostructures, UMR 5586 of CNRS and Univ. Lyon 1, Lyon, France, LAURENT LIMAT, Laboratoire Matiere et Systemes Complexes, UMR 7057 of CNRS and Univ. Paris Diderot, Paris, France — In our experiment, a circular hydraulic jump is formed by a viscous jet impacting a horizontal or slightly tilted glass disk. A drop of the same liquid, deposited in the jump does not coalesce, and remains trapped at its periphery, because of the air entrainment entering suddenly a liquid bath of higher density. This leads to the formation of a liquid sheet floating on the bath, the line of intersection of the substrate and liquid/liquid interface being called the grounding line in literature relative to ice field formation. At long enough time, and for small enough substrate inclination, the position of the grounding line and the angle of detachment of the flow are given by very simple expressions, that are in good agreement with our experiments. This angle of detachment depends only on the substrate inclination and on the relative density mismatch, while the distance between the grounding line and the initial shore position is proportional to the inverse of the inclination angle.
When a wheel of plow is dragged at a constant velocity within a granular medium can experience a vertical force, whose sign and magnitude depend on the shape of the intruder and the depth. Simulations show that the lift as well as drag are generated mainly by interaction with the leading surface. Comparing the stress on flat plates with different inclination angles with the surface stresses on the intruders indicates that shape dependent drag and lift can be understood as the sum of the contributions from differential (flat plate) elements. A model similar to Coulomb’s wedge method is developed to describe the forces experienced by the flat plates.

The lift and drag forces are generated mainly by interaction with the leading surface. Comparing the stress on flat plates with different inclination angles with the surface stresses on the intruders indicates that shape dependent drag and lift can be understood as the sum of the contributions from differential (flat plate) elements. A model similar to Coulomb’s wedge method is developed to describe the forces experienced by the flat plates.

The lift and drag forces are generated mainly by interaction with the leading surface. Comparing the stress on flat plates with different inclination angles with the surface stresses on the intruders indicates that shape dependent drag and lift can be understood as the sum of the contributions from differential (flat plate) elements. A model similar to Coulomb’s wedge method is developed to describe the forces experienced by the flat plates.
3:57PM RN.00005 Wedge model of force and flow oscillations in plowed granular media. PAUL B. UMBANHOWAR, Northwestern University, NICK GRAVISH, DANIEL I. GOLDMAN, Georgia Institute of Technology — We develop a model that captures the changing response of granular media with volume fraction, $\phi$, to a partially submerged vertical plate dragged horizontally at low velocity. In experiment, a bifurcation in force and flow occurs at the onset of grain dilatancy, $\phi_d$. Below $\phi_d$ rapid irregular fluctuations in the drag force, $F_D$, are observed. Above $\phi_d$ fluctuations in $F_D$ are periodic and increase with $\phi$. Velocity field measurements indicate $F_D$ fluctuations are correlated with the creation and destruction of shear bands during drag. Shear bands originate at the base of the plate and extend to the surface forming a nearly triangular wedge of material moving with the plate. Our model assumes that $F_D$ originates in the force required to overcome sliding friction and push the wedge of material up the slope defined by the inclination of the shear band. Combined with the fact that shear bands are weaker (stronger) than the bulk material for $\phi > \phi_c$ ($\phi < \phi_c$) our model quantitatively predicts the observed dependence of $F_D$ fluctuations and flow on time and $\phi$ for $\phi > \phi_c$ and gives significant insight into the non-periodic fluctuations observed for $\phi < \phi_c$.

4:10PM RN.00006 ABSTRACT WITHDRAWN

4:23PM RN.00007 Time-Dependent Continuum and Molecular Dynamics Simulations of Density Inversion in Shaken Granular Layers$^1$. JON BOUGIE, VERONICA POLICHT, Physics Department, Loyola University Chicago, JENNIFER KREFT PEARCE, Department of Chemistry, University of Texas at Tyler — We investigate density inversion in vertically oscillated granular layers using continuum and molecular dynamics simulations. Layers of grains atop a plate that is shaken sinusoidally in the direction of gravity will leave the plate at some time in the cycle if the maximum acceleration of the plate $a_{\text{max}}$ exceeds the acceleration of gravity $g$. For some values of shaking frequency $f$ and accelerational amplitude $a_{\text{max}}$, a small region near the plate displays time-dependence in response to the sinusoidal shaking, while the bulk of the layer reaches a steady-state. In certain cases, the system exhibits a “density inversion” in which a low density granular gas supports a higher density layer of grains. We use three-dimensional simulations of time-dependent continuum equations as well as molecular dynamics simulations to study both the time-dependent and the steady-state regions of the flow.

Tuesday, November 23, 2010 3:05PM - 4:36PM — Session RQ Bubbles V Long Beach Convention Center 203B

3:05PM RN.00001 Bubble distributions and dynamics in the far wake of a ship. DOUGLAS SCHWER, RUSSELL DAHLBURG, JAY BORIS, Naval Research Laboratory — This research focuses on how to simulate efficiently the dynamics of the bubble size distribution in the far wake of a surface ship with a water jet or conventional propulsion system. In this region, the wake is undergoing turbulent decay and the bubble void fraction is low enough to justify one-way coupling. A particle-tracking method is used to create an “ideal” bubbly flow solution within a decaying wake, accounting for buoyancy, agglomeration, dissolution, drag, and convection. These solutions are then compared with multi-group methods for bubble size distributions and dynamics. Multi-group methods are generally much more efficient than particle-tracking methods, but determining appropriate expressions for the different bubble processes described above can be less straightforward. A comparison of the evolution of the bubble distribution in the far wake between the multi-group and particle-tracking methods shows how well the multi-group methods are able to capture the bubble dynamics in these flow regimes.

3:18PM RN.00002 Deformation-induced lateral migration of a bubble slowly rising near a vertical plane wall$^1$. KAZUYASU SUGIYAMA, The Univ. of Tokyo, FUMIO TAKEMURA, AIST — A deformation-induced lateral migration of a nearly spherical bubble rising near a vertical plane wall in a stagnant creeping liquid flow is numerically studied by means of a boundary-fitted finite-difference approach (Sugiyma & Takemura (2010) J. Fluid Mech. accepted). The migration velocity is obtained using Lorentz’s reciprocal theorem as a function of $\epsilon$, corresponding to a ratio of a bubble-wall gap to the bubble radius. For $\epsilon \gg 1$, the simulated migration velocities are consistent with an available analytical solution for the wide-gap case (Magnaudet et al. (2003) J. Fluid Mech. 476, 115). With decreasing $\epsilon$, the lift force is found to be more affected by the high-order deformation modes. The simulation and the lubrication analysis (Hodges et al. (2004) J. Fluid Mech. 512, 95) consistently demonstrate that when $\epsilon \ll 1$, the lubrication effect makes the migration velocity asymptotically $\mu V/(\gamma D^2)$ (here, $V$ is the rising velocity, $D$ the liquid viscosity, and $\gamma$ denote the rising velocity, the liquid viscosity, and the surface tension, respectively). However, the experimentally measured migration velocity is considerably higher by a factor of about 3 than the simulated one, implying that unexpected factors may be involved in the system.

$^1$Supported by the Grant-in-Aid for Young Scientist (B) (No.21760120) of MEXT.

3:31PM RN.00003 Saddle-point dynamics in bubble break-up. LIPENG LAI, WENDY W. ZHANG, University of Chicago — Cylindrically-symmetric bubble break-up are unstable against azimuthal perturbations. While most perturbations preempt the symmetric pinch-off singularity by creating a smooth contact, our simulations also show a qualitatively different, non-monotonic evolution for certain narrow ranges of initial perturbation, the non-monotonic shape evolution proceeds as follows: the neck cross-section collapses into a narrow and long slot shape. The two ends of the “slot” initially sharpen rapidly. Then the sharpening slows. Eventually the curvatures of the two ends invert, creating two narrow fingers of water that intrude into the bubble interior. As time goes on, the tip of the intrusion broadens while the finger remains relatively narrow, causing the entire intrusion to resemble a mushroom on a thin stalk. This sharpen-first-then-broaden sequence is qualitatively consistent with a phase space trajectory controlled by the presence of a saddle point. The maximum end curvature attained during the time evolution appears to diverge as the amplitude or the phase of the initial Fourier mode distortion is tuned towards appropriate threshold values. This suggests that the saddle corresponds to a singular interface shape.

3:44PM RN.00004 Complex fluid pinch-off in bubble rafts$^1$. CHIN-CAHNG KUO, University of California, Irvine, SHERYLL NERY, University of California, Los Angeles, MIKE ARCINAGA, MICHAEL DENNIN, University of California, Irvine, DEPARTMENT OF PHYSICS AND ASTRONOMY TEAM — Pinch-off processes have been investigated in two and three dimensional liquid systems. A common element of pinch-off is the existence of a well-defined scaling regime in which the minimum radius of the system decreases as a power-law in time. The exact value for the power-law depends on the dominant mechanism in the material and the dimensionality. For complex fluids, the dynamics are strongly dependent on the applied stress, rate of strain, and the inner structure of the material, which lead to interesting pinch-off behavior. Here we present the experimental results for pinch-off in bubble rafts pulled by two parallel plates on a liquid surface. Power-law behavior is observed, and we will report on the impact of pulling speed and composition on the value of the power-law exponents.

$^1$We acknowledge to NSF-DMR-0907212 and the Research Corporation.
3:57PM RQ.00005 Pinch-off of axisymmetric squashed underwater bubbles , DANIEL C. HERBST, WENDY W. ZHANG, University of Chicago — Up until now, theoretical and computational studies of bubble pinch-off have assumed for simplification that the neck near break-up is nearly cylindrical, and that the surrounding water flows inwards radially. In this regime, azimuthal perturbations, however small initially, give rise to vibrations that dominate the collapse. Here we use a boundary integral method to investigate the surface evolution starting from initial states in the opposite limit, where the neck shape is composed of two cones with large opening angle. We also compare simulation results near the minimum against predictions from a leading-order model valid in the limit of a weakly deformed neck. The problem is governed by the Weber number, the problem is governed by the Weber number, 

\[ \Lambda = \frac{\rho_w u_w h_a}{\sigma} \]

and a constant, \( \Lambda = 5.52 \). To describe the complex variation observed in the bubbling frequency and the size of the bubbles formed. In addition, direct numerical simulations have been performed by means of the Volume of Fluid technique (VoF), and the results compared with the experimental measurements.

3:18PM RS.00002 Flow characterization in vegetated marsh environments , JENAHVIVE MORGAN, ALINE COTEL, PAUL WEBB, University of North Carolina — The evaluation of wake flows due to aquatic vegetation is necessary to understand the response of the environment to flow through a marsh. Considering the influence of vegetation on the turbulent characteristics of the flow is important in understanding its effect on the surrounding environment and can be applied to the design and creation of artificial marsh environments for restoration projects. Vegetative environments, due to their structure, create turbulence in the flow which in turn affects the response of the native fish species, as well as contaminant and sediment transport. In an effort to model an aquatic vegetative environment, arrays of vertically aligned cylinders of diameter equal to 1/4" were set-up in staggered positions to create a variety of flow configurations in a re-circulating water tunnel. Particle Image Velocimetry (PIV) was used to determine flow characteristics at different velocities for each geometry. In particular, turbulence downstream of the cylinders was examined for different arrangements of the marsh model. The data reveal a strong relationship between the arrangement of the cylinder arrays and the wake turbulence downstream of the cylinders. These results have implications for fish responses to aquatic environment and the design of artificial wetlands.

3:31PM RS.00003 Shell selection of hermit crabs is influenced by fluid drag , BARBARA CASILLAS, RENE LEDESMA, GUILLERMINA ALCARAZ, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — The flow around gastropod shells used by hermit crabs (Calcinus californiensis) was visualized experimentally. These crabs choose their shells according to many factors; we found that the choice of shell (shape and weight) is directly related to the drag caused over them by the exposure to wave action. Tests were conducted in a wind tunnel to investigate flow differences for shells of various shapes. A particle image velocimetry (PIV) system was used to visualize the flow field. The images above show the flow field around two types of shells (Thais speciosa and Nerita scabircosta) for Reynolds numbers of O(10^5). Using a control volume analysis, the drag coefficient was inferred. Several shell geometries, orientations and mean flow velocities were tested. In this talk, the flow and drag force will be shown for the different arrangements. A discussion of the relation between drag and shape will be presented.
3:44PM RS.00004 Numerical simulation on the assimilation of substrate by microorganisms in a turbulent flow. MARION LINKES, PASCAL FEDE, IMFT, JEROME MORCHAIN, PHILIPPE SCHMITZ, LISBP — A strong decrease in the conversion yield of substrate into biomass is constantly observed in fed-batch bioreactors when passing from a laboratory to an industrial scale because of concentration gradients that influence the biomass behaviour. In this work, the focus was emphasised on the effect of the mixing on the microorganisms. This was addressed through a one-dimensional diffusion model for the transport of substrate towards the cell with specific boundary conditions at the microorganism, that plans to simulate its limiting behaviour. Diversified far field concentrations in substrate were investigated and aimed to mimic the several states of mixing at the small scale of the flow. The influence of relevant parameters of the entering substrate concentration was scrutinised in term of interfacial response at the cell interface. This interfacial response takes into account the concentration and assimilation rate at the microorganism interface, and its analysis shows results in good agreement with different experimental observations and the cell affinity for the substrate has been demonstrated taking the mixing state of the latter.

3:57PM RS.00005 Power generation by flagella-propelled Serratia Marcescens†, TRUNG-HIEU TRAN, Department of Aerospace and Information Engineering, Konkuk University, Seoul, Korea, MIN JUN KIM, Department of Mechanical Engineering, Drexel University, Philadelphia, Pennsylvania, USA, DOYOUNG BYUN, Department of Aerospace and Information Engineering, Konkuk University, Seoul, Korea — In this study, we present electrical power generation by using swimming Serratia marcescens which is a rod shaped bacterium species and has about 10 um long and about 20 nm thin helical filaments. Flow in micro channel is driven by bacteria attached on the wall, which is around 25 to 50 µm/sec. The driven electrolyte solution flow (buffer solution containing high concentration of S. marcescens) may be considered as movement of conductor. If we place permanent magnets on the top and bottom of the micro channel and electrodes on side walls in the micro channel, electrical current could be generated by the principle of Lorentz force acting on the moving charges. The potential between the two electrodes was measured to be up to 10mV and the electrical current was about 10pA with external load 50 Ohm. Even if the energy generated by bacteria swimming is small, it demonstrated the possible generation of power, which requires in-depth further research.

4:10PM RS.00006 ABSTRACT WITHDRAWN — Tuesday, November 23, 2010 3:05PM 4:49PM Session RT Bioluminocom XII: Macro-Swimming III Long Beach Convention Center Grand Ballroom B

3:05PM RT.00001 Flexibility and Resonance in Thrust Production of a Mechanical Swimming Lamprey. MEGAN LEFTWICH, ALEXANDER SMITS, Princeton University, PRINCETON UNIVERSITY TEAM — We use a robotic lamprey as a means of investigating the influence of flexibility on the wake structure and thrust production during anguilliform swimming. A programmable microcomputer actuates 11 servomotors that produce a traveling wave along the length of the lamprey body. The waveform is based on kinematic studies of living lamprey. The shape of the tail is taken from CT scan data of the silver lamprey, and it is constructed of flexible PVC gel. Plastic inserts allow the the degree of flexibility to be changed. PIV measurements in the wake of the robot with three different flexible tails show that a 2P structure dominates the flexible wake. However, the large structure is composed of several small vortices (as opposed to the large coherent vortex seen behind a stiff tailed robot). Furthermore, the wake loses coherence as flexibility is increased. Additionally, momentum balance calculations indicate that increasing the tail flexibility yields less thrust. Finally, we find that changing the cycle frequency to match the resonance frequency of the tail increases the thrust production. The project is supported by NIH CNRS Grant 1R01NS054271.

3:18PM RT.00002 How does muscle forcing lead to translational motion in undulatory swimming?†, AMNEET BHALLA, NEELESH PATANKAR, Northwestern University — Swimming organisms show variety of complex deformations during their movement. In this work we enquire whether complex muscle forcing is required to create the observed deformation kinematics that cause movement. We interrogate how the muscle forcing leads to the forward translational momentum of an organism. A set of linearized equations of motion, using a spring-link model, is derived for undulatory swimming. We do not consider observed body deformations to be composed of active and passive components. Instead, swimming is treated as a forced oscillation problem. Forcing can be due to the muscles (active swimming) or due to the surrounding fluid (passive swimming). In either case, the forcing triggers the first few fundamental deformation modes of the body which in turn drive the axial translational motion. We explain the reason for observing only the first few fundamental modes. It is seen that simple forcing patterns can trigger complex looking deformation kinematics that lead to movement. We show that there is range of frequency at which the body responds well (i.e. the swimming speed increases with frequency), but after that range the body does not respond well to higher frequencies. It is found, consistent with prior work, that anisotropy in drag enables swimming.

3:31PM RT.00003 Stability of Passive Locomotion in a Perfect Fluid†, FANGXU JING, EVA KANSO, University of Southern California — We investigate the effect of body elasticity on the stability of locomotion in a perfect fluid. Our motivation is to study fish swimming. Actual fish seem to alternate between actively flapping and passively responding to the surrounding fluid, referred to as Burst and Coast cycle. We study the stability of the coast (passive) phase. It’s well known that the passive motion of a single elongated rigid body along its major axis of symmetry is unstable. The question is: can passive changes mediated by body elasticity stabilize the motion? The answer is yes. We consider an articulated body with finite number of rigid links, connected by hinge joints with torsional springs at the joints to emulate the elasticity of fish. The motion of the articulated body with constant velocity along its major axis of symmetry is a relative equilibrium. Upon analyzing the stability of this equilibrium, we discover that passive shape changes do stabilize the motion for appropriate combination of body geometry and spring stiffness. We plot the region of stability in aspect ratio - spring stiffness parameter space.

3:44PM RT.00004 A self-excited flapper from fluid-structure interaction. OSCAR M. CURET, KENNETH S. BREUER, Brown University — The flexible nature of lifting and propulsive surfaces is a common characteristic of aquatic and aerial locomotion in animals. These surfaces may not only move actively, but also passively or with a combination of both. What is the nature of this passive movement? What is the role of this passive motion on the force generation, efficiency and muscle control? Here, we present results using a simple wing model with two degrees of freedom designed to study passive flapping, and fluid-structure interaction. The wing is composed of a flat plate with a hinged trailing flap. The wing is cantilevered to the main body to enable a flapping motion with a well-defined natural frequency. We test the wing model in a wind tunnel. At low speed the wing is stationary. Above a critical velocity the trailing wing section starts to oscillate, generating an oscillating lift force on the wing. This oscillating lift force results on a self-excited flapping motion of the wing. We measure the kinematics and the forces generated by the wing as a function of flow velocity and stiffness of the cantilever. Comparisons with aeroelasticity theory will be presented as well as details of the fluid-structure interactions.
Recent studies have indicated the importance of structural properties to the hydroelastic response of passive flexible bodies in uniform flow. One response regime includes a structural traveling wave of increasing amplitude from leading to trailing edge with alternating vortex shedding in the wake. This modal response exhibits the same characteristics as fish swimming, suggesting the importance of the natural hydroelastic response in fish swimming actuation. We explore the concept of underactuation in fish swimming by examining the ability to achieve swimming kinematics through single point forcing of a flexible body. The phenomenon is first studied through simulation of the Navier-Stokes equations coupled to a nonlinear structural solver. This indicates the relationship between passive and active modal response, and the ability to alter the vortex wake and associated hydrodynamic loading through underactuation. A reduced-fidelity model for the fluid-structural dynamics is employed to optimize the properties of a fish body for the desired underactuated modal response. The optimized design is then tested in a captive-swimming experiment to examine the response modes and swimming performance.

1 Funding under NSF CBET grant 0932352 is gratefully acknowledged.

4:36PM RT.00008 Examination of Scuba Fin Designs Using Simultaneous Force and DPIV Measurements, LORI HALVORSON, ERICA SHERMAN, CHIAMIN LEONG, TIMOTHY WEI, RPI — Like many commercial products, there is a wide variety of scuba dive fins on the market, each one of which, the designers argue are the best and most efficient. The foundation for these claims invariably are based on a sort of hydrodynamic argument with the full spectrum of scientific credibility attached. In this study, we examine a number of commercially available scuba fins using both DPIV of the fin motion as well as dynamic force measurements of thrust generated by a swimmer kicking against a stationary force balance. Both techniques have been used and reported in the past for studies of world class swimmers and dolphins. This will be the first time that high quality data has been obtained for both fin movement and force simultaneously. A number of different fin designs were tested. But the most interesting comparison was between the "monofin" and the "split-fin" designs. A discussion of the relative merits of the two different designs will be presented along with video footage showing flow and force overlaid on the fin motions.

Tuesday, November 23, 2010 3:05PM - 4:36PM
Session RU Particle Laden Flows III Hyatt Regency Long Beach Regency A

3:05PM RU.00001 Size distribution of droplets undergoing phase transition in homogeneous isotropic turbulence, BRITI SUNDAR DEB, BERNARD J. GEURTS, HERMAN J.H. CLERCKX, A.K. KUCZAJ, HANS KUERTEN, Multiscale Modeling and Simulation, Faculty EEMCS, University of Twente, The Netherlands — We investigate the dynamics of an ensemble of discrete aerosol water droplets undergoing phase transition, expressed by evaporation and condensation, in a turbulent flow. Our focus is on the stationary distribution of droplet sizes that develops as a result of these phase transitions in forced, homogeneous, isotropic turbulence. For this purpose we perform direct numerical simulation (DNS) using a de-aliased pseudo spectral method in a domain with periodic boundary conditions. We solve the Navier-Stokes equations and additional equations for the temperature and background humidity against which the size of the droplets evolves by exchanging heat and mass. The motion of the droplets under Stokes drag force is time-accurately tracked. The responsiveness of the droplets to small turbulent scales is directly related to the size of the individual spherical droplets. The latter is changing due to evaporation and condensation, which in turn depends on the unique trajectory of the droplets in the unsteady flow. We compute the natural size distribution at various heat and mass transfer parameters and observe its dependency on the Reynolds number.

3:18PM RU.00002 Multiple Particle Interaction at Intermediate Reynolds Numbers, EL YACOUBI, Cornell University, SHENG XU, Southern Methodist University, Z. JANE WANG, Cornell University, SHENG XU COLLABORATION — The literature is rich with studies on particle-particle interaction in Stokes flow. However, there are scant studies on particle interaction at intermediate Reynolds numbers. Here, we present a new computational scheme to simulate the dynamics of the particles coupled to the Navier-Stokes solutions for the fluid. In order to understand the basic picture of particle-particle interactions in fluid, we investigate the dynamics of an array of freely falling cylinders with an initial spacing on the order of the particle diameter. We find that for a small number of particles (n = 3, 4), there are two distinct falling configurations which depend on the parity of n. For n > 4, the falling configuration is a mix of those previous modes. However, when the initial spacing between particles is below a threshold, the array is separated into small clusters of 2 or 3 particles. We further quantify the interaction force between two falling particles as a function of their relative position, and compare them with results in the Stokes regime.
3:31PM RU.00003 Possibility of perfect fluid flow from granular jet impact, JAKE ELLOWITZ, NICHOLAS GUTTENBERG, HERVE TUR LiER, WEN D Y W. ZHANG, SIDNEY R. N AGEL, University of Chicago — Axisymmetric collision of a cylindrical water jet with a circular target generates a thin conical sheet, also known as a water bell [1]. Intriguingly, recent experiments on granular jet impact in the regime of dense inertial flow reveal similar behavior: the angles by which the collimated sheets of particles are ejected from the target [2] agree closely with the angles measured in the water-bell experiments. This quantitative correspondence suggests that the collective granular motion during impact can be modeled as an incompressible, continuum fluid. Since viscous effects are weak in water-jet impact and the granular jet is comprised of non-cohesive particles (hence possessing zero surface tension), the simplest scenario is that the continuum motion corresponds to the flow of a perfect fluid. We assess this possibility by comparing exact solutions of 2D Euler-jet impact with 2D discrete-particle simulations of granular impact. We also construct approximate solutions for axisymmetric Euler-jet impact and compare these with granular-impact experiments.


3:44PM RU.00004 Gas and particulate phase velocity measurements of a high-speed gas jet into a two-dimensional bubbling fluidized bed, ALEXANDER MYCH KOVSKY, STEVEN CECCIO, University of Michigan — A Laser Doppler Velocimetry (LDV) technique was implemented to simultaneously measure the gas and particulate phase velocities in a high-speed jet plume in a two-dimensional (2D) bubbling fluidized bed. The gas and particulate phase velocity profiles are presented and analyzed. This includes similarity profile scaling as well as volume fraction, mass flow, and momentum transport calculations for the two phases. Furthermore, applying the Eulerian equation of motion to the particulate phase with the measured velocity profiles, the bed particle drag coefficient is recovered and is found to be consistent with the established empirical value.

3:57PM RU.00005 An experimental technique for simultaneous measurement of fluid flow and particle kinematics in particle-laden flows, AUDRIC COLLIGNON, EVAN VARIANO, UC Berkeley, Civil & Environmental Engineering Dpt — A significant challenge facing laboratory measurements of particle-laden flow is the simultaneous resolution of both fluid and particle phases. We present a simple technique to resolve the kinematics of individual particles and the surrounding flow. Most importantly, this technique reveals the full angular velocity vector of each particle. We use water as our fluid (allowing high Reynolds number flow) and particles that have the same refractive index as water. Results from spherical hydrogel particles will be presented and other options will be discussed. Refractive index matching allows light to propagate undisturbed through particles, even at high volume loading. We apply PIV simultaneously to the fluid phase and to the interior of each particle. We then use the velocities measured inside each particle to solve an inverse problem given particle location, translation, and angular velocity. We present the technique, including details of the optical setup and image processing methods. We also present a validation and uncertainty analysis covering random and bias errors.

4:10PM RU.00006 Finite-size particles in turbulence: effect of particle shape and rotational dynamics, GABRIELE BELLANI, KTH Mechanics, Stockholm, Sweden, EVAN A. VARIANO, UC Berkeley, CEE department, Berkeley (CA), USA — In this laboratory study we investigate the two-way coupling between rigid particles and homogeneous isotropic turbulence. Turbulence Reynolds number is $Re_t \approx 350$ and the particle length and time scales are within the inertial sub-range. We focus on the effects of particle shape and rotation. Rotational dynamics play an important role in inter-phase momentum exchange, especially for non-spherical particles. A novel technique resolves particle velocity and rotation, simultaneously with fluid-phase velocities. From these measurements we analyze the inter-phase coupling of both translational and rotational motion. Analysis includes correlations between particle motion and the surrounding fluid, wake dynamics, and particle motion statistics. Effect of particles on the turbulent flow is investigated from the fluid-phase turbulent kinetic energy and dissipation rates.

3:31PM RU.00003 Probing particle transport in closed-streamline flows with microfluidic devices, SHAHAB SHOJAEE-ZADEH, Rutgers University, JEFFREY MORRIS, The City College of New York — We use microfluidic devices to study the flow of neutrally buoyant suspension around bluff-bodies. We use low-viscosity liquids and monodisperse particles of diameter below 10 μm at a constant concentration of 8.4 volume %. Several bluff-body geometries are introduced and by using high-speed video imaging we observe a striking segregation of the particles and fluid in the wake region at elevated Reynolds numbers. Based on 2-D and 3-D flow field simulations, we interpret the migration of particles and their trajectories across the streamlines based on the geometry of the bluff-body. Experimental observations reveal that if particles are forced into an initially particle-depleted region, they will eventually leave and will bring the system to its original state.

Tuesday, November 23, 2010 3:05PM - 4:23PM — Session RV Suspensions III — Hyatt Regency Long Beach Regency B

3:05PM RU.00001 A falling cloud of particles at small but finite Reynolds number, FLORENT PIGNATEL, MAXIME NICOLAS, ELISABETH GUAZZELLI, IUSTI-CNRS UMR 6596, Polytech-Marseille, Aix-Marseille Université (U1), GEP TEAM — Through a comparison between experiments and numerical simulations, we have examined the dynamics of a cloud of spheres at small but finite Reynolds number. The cloud is seen to flatten and to transition into a torus which further widens and eventually breaks up into droplets. While this behaviour bears some similarity with that observed at zero-inertia, the underlying physical mechanisms differ. Moreover, the evolution of the cloud deformation is accelerated as inertia is increased. Two inertial regimes where macro-scale inertia and micro-scale inertia become successively dominant are clearly identified.

3:18PM RU.00002 Sedimentation and Effective Temperature of Active Colloidal Suspensions, JEREMIE PALACCI, CECILE COTTIN-BIZONNE, CHRISTOPHE YBERT, LYDERIC BOCQUET, LPMCN-Universite de Lyon, LPMCN-UNIVERSITE DE LYON TEAM — We investigate experimentally the non-equilibrium steady state of an active colloidal suspension under gravity field. The active particles are made of chemically powered colloids, showing self propulsion in the presence of an added fuel, here hydrogen peroxide. The active suspension is studied in a dedicated microfluidic device, made of permeable gel microstructures. Both the microdynamics of individual colloids and the global stationary state of the suspension under gravity field are measured with optical microscopy. This yields a direct measurement of the effective temperature of the active system as a function of the particle activity, on the basis of the fluctuation-dissipation relationship. Our work is a first step in the experimental exploration of the out-of-equilibrium properties of active colloidal systems. Working along this line, we also present first signatures of collective properties in active suspensions.

3:31PM RU.00003 Probing particle transport in closed-streamline flows with microfluidic devices, SHAHAB SHOJAEE-ZADEH, Rutgers University, JEFFREY MORRIS, The City College of New York — We use microfluidic devices to study the flow of neutrally buoyant suspension around bluff-bodies. We use low-viscosity liquids and monodisperse particles of diameter below 10 μm at a constant concentration of 8.4 volume %. Several bluff-body geometries are introduced and by using high-speed video imaging we observe a striking segregation of the particles and fluid in the wake region at elevated Reynolds numbers. Based on 2-D and 3-D flow field simulations, we interpret the migration of particles and their trajectories across the streamlines based on the geometry of the bluff-body. Experimental observations reveal that if particles are forced into an initially particle-depleted region, they will eventually leave and will bring the system to its original state.
3:44PM RV.00004 Fluid Mechanics of Cellulose Fiber Suspensions Using MRI.** ROBERT POWELL, DAVID LAVENSON, EMILIO TOZZI, University of California Davis, MICHAEL MCCARTHY, University of California — Efficient processing of fibrous biomass requires understanding the mechanics of fiber suspensions having large particle sizes of biomass particles, fast settling, and entanglements. Direct imaging of velocity profiles using magnetic resonance imaging provides a way of characterizing flow in the presence of such non-idealities. We found a strong influence of fiber length, concentrations and flow rates on velocity profiles and pressure drops. We map different regions in the concentration-velocity plane that serve as a guide to decide whether or not to use generalized newtonian rheological models. The concentration effects were best described by the use of a crowding number, with large changes in pressure and velocity profiles occurring in a narrow range of crowding numbers. Qualitative differences between the behavior of the long fibers and the short and medium fibers demonstrate a strong effect of fiber aspect ratio on rheology.

3:57PM RV.00005 The suspension balance model revisited1. ** PRABHU NOTT, Indian Institute of Science, ELISABETH GUAZZELLI, OLIBER POULIQUEN, IUSTI - Polytech’ Marseille — This paper addresses a fundamental discrepancy between the suspension balance model and other two-phase flow formulations. The former was proposed to capture the shear-induced migration of particles in Stokesian suspensions, and hinges on the presence of a particle phase stress to drive particle migration. This stress is taken to be the "particle stress", defined as the particle contribution to the suspension stress. On the other hand, the two-phase flow equations derived in several studies show only an average force acting on the particle phase, but no stress. We show that the identification of the particle phase stress with the particle stress in the suspension balance model is incorrect, but there exists a well-defined particle phase stress. Following the rigorous method of volume averaging, we show that the average force on the particle phase may be written as the sum of an inter-phase force and the divergence of the particle phase stress. We derive exact relations for these quantities. We also comment on the interpretations and results of previous studies that are based on the identification of the particle phase stress with the particle stress.

4:10PM RV.00006 Coupling Between Translational and Orientational Ordering in Fiber Suspensions1, ALEXANDRE FRANCESCHINI, EMMANOUELA FILIPPIDI, Center for Soft Matter Research, New York University, ELISABETH GUAZZELLI, IUSTI-CNRS UMR 6595 - Polytech Marseille - Aix-Marseille Université, DAVID PINE, Center for Soft Matter Research, New York University — Suspensions of non-colloidal particles under slow periodic strain undergo a non-equilibrium dynamical phase transition from an absorbing state to an active fluctuating state. In the case of spherical particles, this critical absorbing-phase transition is observed at a single strain amplitude. In the case of rod-like particles, the transition between fluctuating and absorbing phase is observed over a continuous range of applied strain amplitude. Orientational degrees of freedom couple to translational degrees of freedom, expanding the critical domain from a point to a line. Experiments and calculations show the orientation distribution of the rods with time and its relation with respect to the critical strains. Power-law relaxations are observed close to criticality and the measured exponents are consistent with Manna universality class of directed percolation models.

Tuesday, November 23, 2010 3:05PM - 4:36PM — Session RW Experimental Techniques IV Hyatt Regency Long Beach Regency C

3:05PM RW.00001 Ultrasound Doppler Velocimetry Measurements in Turbulent Liquid Metal Channel Flow1, MICHEL RIVERO, CIE-UNAM, DANDAN JIAN, CHRISTIAN KARCHER, TU-Ilmenau, Germany, SERGIO CUEVAS, CIE-UNAM — Control of molten metal flow using magnetic fields is important in industrial applications. The Electromagnetic Flow Control Channel (EFCO) is an experimental test facility, located at Ilmenau University of Technology, for the development of such kind of control systems. The working fluid is the low-melting liquid metal alloy GaInSn in eutectic composition. In this channel, flow control is realized by combining and coupling the non-contact flow driving technology of electromagnetic pumps based on rotating permanent magnets and the non-contact flow rate measurement technology termed Lorentz Force Velocimetry (LFV).

The flow rate is adjusted by controlling the rotation rate of the permanent magnet system. Physically, LFV is based on measuring the force acting on a magnet system. This force is induced by the melt flow passing through the static magnetic field generated by the system and is proportional to the flow. To calibrate the flow rate, we apply UDV technique to measure and analyse both turbulent hydrodynamic and MHD flow profiles in EFCO at various Reynolds numbers.

1 Acknowledge support by BMBF within the ForMaT program and DFG within the Research Training Group Lorentz force and to CONACYT.

3:18PM RW.00002 Evaluation of a Penning Mixture for Use in Plasma Adaptive Optics. ** BRIAN NEISWANDER, ERIC MATLIS, THOMAS CORKE, University of Notre Dame — This research examines the use of a Penning gas mixture in a dielectric barrier discharge (DBD) plasma device to achieve stable plasmas at high pressures. Previous research suggests increasing pressure produces a larger dynamic range of refractive index, which is favorable for adaptive optics. As the pressure increases, however, plasma generation in air soon becomes impractical due to power requirements. Penning mixtures, such as neon with a small amount of argon, feature lower breakdown voltages and stronger ionization attributed to the presence of Penning ionization. Experimental measurements of voltage, current, power, and electron density are presented for a DBD plasma chamber containing the gas mixture. Results are evaluated against a previously developed empirical model. The work further motivates the creation of a plasma adaptive optics system.

3:31PM RW.00003 High-precision image-based tracking of a rigid body moving within a fluid, STUART LAURENCE, JAN MARTINEZ-SCHRAMM — Precise measurement of the displacement, velocity and acceleration of a moving rigid body is of interest in many applications. The use of imaging techniques to obtain such information is an attractive option, particularly for movement within a fluid, as such measurements are inherently non-intrusive. Here we describe a class of imaging techniques based on edge detection and least-squares fitting for determining the displacement of a body, from which velocities and accelerations are readily derived. The use of edge-detection allows for potentially higher precision than correlation-based techniques. The accuracy of the techniques is estimated using both artificially generated images and calibrated measurements, and displacement errors of the order of a few thousandths of a pixel are shown to be obtainable for camera noise levels of a few percent. The resulting uncertainties in velocity and acceleration measurements are also analyzed. Several applications are then described, with particular emphasis on force measurements in short duration supersonic and hypersonic wind tunnels.
3:44PM RW.00004 Quantification of the Transient Behavior of Wind-driven Water Droplets and Rivulet Flows on a Substrate

3:57PM RW.00005 Wind tunnel experiment on investigating the times of ventilation in case of pollutant dispersion in an urban area

4:10PM RW.00006 Assessment of image correlation methods for the estimation of volume flow rates of subsea oil-gas plumes

4:23PM RW.00007 A Novel 3 Dimension 3 Component Micro-PIV System

Tuesday, November 23, 2010 3:05PM - 4:49PM 
Session RX Material Processing Flows Hyatt Regency Long Beach Regency D

3:05PM RW.00001 Structures Formed by Front Induced Phase-Separation

3:18PM RX.00002 Exploration of novel composite microstructured fibers from capillary instability

3:31PM RX.00003 A Multi-Scale Computer Model for Simulating Polymeric Foaming Processes
3:44PM RX.00004 Simulating the melt blowing of viscoelastic materials, CHUNFENG ZHOU, DAWUD H. TAN, SATISH KUMAR, CHRISTOPHER W. MÁCOSKO, FRANK S. BATES, University of Minnesota — This work is motivated by recent experimental developments in melt blowing that enable the production of nanofibers. In contrast to electrospinning, which is another method for producing nanofibers, melt blowing is potentially faster and environmentally friendlier. Using a slender-jet approximation, we obtain a set of one-dimensional equations governing the fiber area, centerline velocity, and temperature. The upper convected Maxwell (UCM) model and the Phan-Thien and Tanner (PTT) model are used to describe the viscoelastic rheology of the melts. Key to melt blowing is the shear stress on the fiber surface from the external air flow that attenuates the fiber to small diameter. Larger shear stresses or higher air flowrates produce fibers with smaller diameter. Our results show a significant influence of viscoelasticity on melt blowing, especially on fiber diameter. The fiber diameter is found to increase with polymer elasticity, which agrees qualitatively with experimental observations.

3:57PM RX.00005 Zero leakage sealings

BERNARD KOTESOVEC, HERBERT STEINRÜCK, Technical University Vienna — The piston rod of a reciprocating compressor is sealed with elastic cylindrical sealing elements. Across the sealings the pressure drops from the operating pressure to the ambient pressure. The lubrication gap between the elastic sealing and reciprocating piston rod is studied with the aim to find conditions of a leakage free sealing. The flow in the lubrication gap and the elastic deformation of the sealing are determined simultaneously. The net-flow during one cycle of the reciprocating piston rod is calculated. It turns out that maintaining zero leakage is very sensitive. Indeed the outbound flow during out-stroke has to be equal the inbound flow during the in-stroke. By prescribing a special shape of the undeformed sealing zero leakage can be attained — at least theoretically for certain operating conditions. It turns out that temperature dependent material data and a model for cavitation is necessary. The model, its numerical implementation and results will be discussed.

1This work has been financed by the company HOERBIGER VENTILWERKE GmbH & Co KG.

4:10PM RX.00006 Numerical Analysis on Oil Leakage of Fluid Dynamic Bearing for External Impact Test, SUNGHOON BAEK, SIMON SONG, Hanyang University — We conducted numerical simulations on the behaviors of lubricant in the fluid dynamic bearing of a mini motor shocked by external impact using a commercial software. Numerical studies on the behaviors are necessary because it is very difficult to observe the behaviors of lubricant oil and air interface in experiments although the oil leakage have to be prevented for a mini motor used for hard disk drive. To investigate the behaviors of a free surface between lubricating oil and air in the bearing, an unsteady volume-of-fluid model was utilized as well as a Navier-Stokes equation solver. Also, hybrid meshes were adapted: unstructured grids were generated in the most of large and complex geometric regions while structured grids were used in the small regions of very thin gap (a few microns) between rotor and stator. In addition, dynamic mesh and sliding mesh techniques were employed for the stable dynamic deformation of meshes corresponding to the motion of the rotor due to the impact. The results show that an oil break-up doesn’t occur at the first period of an impact of 1000 ~ 1900G along the rotor axis but it occurs in consecutive periods of 1800G. This presentation will include the effects of Weber number on the oil break-up as well as the numerical results in detail.

4:23PM RX.00007 Thermo-Rheometric Studies of New Class Ionic Liquid Lubricants, SAYAVUR BAKHTIYAROV, New Mexico Institute of Mining and Technology, KENNETH STREET, DANIEL SCHEIMAN, NASA GRC, ALAN VAN DYKE, Case Western Reserve University — Due to their specific properties, such as small volatility, nonflammability, extreme thermal stability, low melting point, wide liquid range, and good miscibility with organic materials, ionic liquids attracted particular interest in various industrial processes. Recently, the unique properties of ionic liquids caught the attention of space tribologists. The traditional lubricating materials used in space have limited lifetimes in vacuum due to the catalytic degradation on metal surfaces, high vaporization at high temperatures, dewetting, and other disadvantages. The lubricants for the space applications must have vacuum stability, high viscosity, low creep tendency, good elastohydrodynamic and boundary lubrication properties, radiation atomic oxygen resistance, optical or infrared transparency. Unfortunately, the properties such as heat flow, heat capacity, thermogravimetric weight loss, and non-linearity in the rheological behavior of the lubricants are not studied well for newly developed systems. These properties are crucial to analyzing thermodynamic and energy dissipative aspects of the lubrication process. In this paper we will present the rheological and heat and mass transfer measurements for the ionic liquid lubricants, their mixtures with and without additive.

4:36PM RX.00008 3D Numerical Simulations of Vacuum Arc Remelting (VAR) Processes, ONIKAR SAHNI, PECOS/ICES, The University of Texas at Austin, ROBERT MOSER, The University of Texas at Austin — The metallurgical structure of superalloys refined by Vacuum Arc Remelting (VAR) is determined by the behavior of the liquid metal pool that exists at the top of the ingot, which is in turn affected by fluid dynamics, heat transfer, electromagnetics and solidification. In this study, we examine the behavior of the liquid metal pool by constructing a coupled multi-physics model of the processes, and performing 3-dimensional transient simulations. Moreover, through complex coupling and boundary models we account for phenomena observed in industrial experiments including localized electric current density, arc meandering, shrinkage of solid ingot, etc. Of interest in these simulations are the effects of variations in heat influx, electric current supply, and external magnetic field on the pool dynamics and ultimately the quality of the ingot produced.

Tuesday, November 23, 2010 3:05PM - 4:36PM
Session RY Instability: Interfacial and Thin Film VII
Hyatt Regency Long Beach Regency E
3:05PM RY.00001 ABSTRACT WITHDRAWN

3:18PM RY.00002 Mixed mode buoyancy-driven instability in a Hele-Shaw cell, J. CARBALLIDO-LANDEIRA, P.M.J. TREVELYAN, C. ALMARCHA, A. DE WIT, NLPC, Universite Libre de Bruxelles, Brussels, Belgium — Buoyancy-driven instabilities of a horizontal interface between two different miscible solutions contained in a Hele-Shaw cell are studied both theoretically and experimentally. Our regime of interest is focused on the case where the fastest diffusing species is located in the upper layer. If the upper solution is denser, a Rayleigh-Taylor (RT) instability develops characterized by a deformation of the interface into fingers. If, on the contrary, the denser solution is on the bottom, a Diffusive Layer Convection (DLC) instability is obtained because of differential diffusion effects. Indeed the fast diffusion downwards of the solute initially contained in the upper zone leads to a depletion and an accumulation zone respectively above and below the contact line where locally convection is triggered. In between these two regimes, a mixed mode dynamics intermediate between the RT and DLC regimes is obtained when the density profiles contain a locally stratifically stable region near the contact line where locally convection is triggered. Experiments show that such an instability generates new plume-like structures around the interface.
Viscous fingering, the phenomenon associated with the Saffman-Taylor instability, occurs when a low viscosity fluid penetrates a fluid of higher viscosity. Structure, depending on the viscosity contrast and injection rate, the model is a system of two partial differential equations: a nonlinear, fourth-order equation for the transport of the order parameter, and an elliptic equation for the pressure of the mixture. The interface thickness is maintained through a double-well bulk potential. We present numerical simulations and a linear stability analysis of the model. Continuum modeling of wetting phenomena is necessary in many scientific and engineering applications, from microfluidics and multiphase flow, to flow and transport in permeable media. The present model is also the first step towards an extended model of multiphase displacements in porous media.

In the absence of surface tension, we derive a generalized Polubarinova-Galin equation in the complex plane, which includes the inertial effects for a circular transition is sufficiently abrupt that no gradual transition has been observed. When there is no metal ion in the less viscous solution (non-reactive case), the transition was never observed. These results are similar to those obtained in a Hele-Shaw experiment using an associating polymer solution (Zhao & Maher, Phys, Rev, E, 47, 4728, (1993)). We have measured the rheological property of the gel by means of a rheometer and investigated the relationship between the observed fingering-fracturing transition and the measured rheological property. Finally, we discuss the similarity between the present result and the result obtained by Zhao & Maher.

Capillary instability driven by a permeability gradient.

This work is supported by NSF, CAPES and a Fulbright fellowship.

Capillary instability driven by a permeability gradient.

3:31PM RY.00003 Capillary instability driven by a permeability gradient. TALAL AL-HOUSEIN, Department of Chemical and Biological Engineering, Princeton University; JESUS HERNANDEZ, Department of Physics, California State University at Northridge; JEFFREY ARISTOFF, Department of Mechanical and Aerospace Engineering, Princeton University; SUZIE PROTIERIE, CNRS - Institut Jean Le Rond d’Alembert; HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University. COMPLEX FLUIDS LAB - PRINCETON UNIVERSITY TEAM - Viscous fingering, the phenomenon associated with the Saffman-Taylor instability, occurs when a low viscosity fluid penetrates a fluid of higher viscosity. This capillary instability is demonstrated in a microfluidic setup composed of two symmetric channels that linearly increase in width, rather than the traditional Hele-Shaw cell. The conditions necessary to achieve this instability are studied. In particular, we determined a critical capillary number below which the instability occurs. The effect of viscosity ratio and permeability gradient are also examined.

3:44PM RY.00004 Inertial effects on viscous fingering in the complex plane. ANDONG HE, ANDREW BELMONTE, The W. G. Pritchard Laboratories, Department of Mathematics, Penn State University. We present the nonlinear unsteady Darcy’s equation, which includes inertial effects for flows in a porous medium or Hele-Shaw cell, and discuss the conditions under which it reduces to the classical Darcy’s law. The simulations exhibit the formation of individual, quasisteady fingers whose properties are characterized as a function of the viscosity ratio and the Peclet number. We show that inertia always has a tendency to stabilize the interface, regardless of whether a less viscous fluid is displacing a more viscous fluid or vice versa.

3:57PM RY.00005 Pattern transition from fingering to fracturing in a reacting Hele-Shaw flow. TOMOHIRO UJIIE, YUICHIRO NAGATSU, MITSUMASA BAN, YOSHIHITO KATO, YUTAKA TADA, Nagoya Institute of Technology, Japan. We have experimentally investigated pattern formation observed when a more viscous aqueous polymer solution is displaced by a less viscous solution including a metal ion in a Hele-Shaw cell. When the two liquids contact, a chemical reaction takes place and a gel is formed. For some concentrations of the polymer and the metal ion, a transition from fingering pattern to fracturing pattern is demonstrated as the injection rate exceeds threshold value. The fingering-fracturing transition is sufficient to place that no gradual transition has been observed. When there is no metal ion in the less viscous solution (non-reactive case), the transition was never observed. These results are similar to those obtained in a Hele-Shaw experiment using an associating polymer solution (Zhao & Maher, Phys, Rev, E, 47, 4728, (1993)). We have measured the rheological property of the gel by means of a rheometer and investigated the relationship between the observed fingering-fracturing transition and the measured rheological property. Finally, we discuss the similarity between the present result and the result obtained by Zhao & Maher.

4:10PM RY.00006 Phase-field modeling of viscous fingering in a Hele-Shaw cell. LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology. When a viscous fluid is displaced by a less viscous one in the gap between two parallel plates, or Hele-Shaw cell, the interface between the two fluids is unstable. For low injection rates the system evolves towards a single channel, known as the Saffman-Taylor finger, while for high rates the interface forms a complicated, branched pattern. Here we present a phase-field model for two-phase displacements that captures the viscous instability. The model reproduces the transition from stable displacement to the Saffman-Taylor finger, and from the latter to a disordered state. Structure, depending on the viscosity contrast and injection rate, the model is a system of two partial differential equations: a nonlinear, fourth-order equation for the transport of the order parameter, and an elliptic equation for the pressure of the mixture. The interface thickness is maintained through a double-well bulk potential. We present numerical simulations and a linear stability analysis of the model. Continuum modeling of wetting phenomena is necessary in many scientific and engineering applications, from microfluidics and multiphase flow, to flow and transport in permeable media. The present model is also the first step towards an extended model of multiphase displacements in porous media.

4:23PM RY.00007 Unstable miscible displacements in Hele-Shaw cells: Three-dimensional Navier-Stokes simulations. RAFAEL OLIVEIRA, ECKART MEIBURG, UC Santa Barbara. We simulate unstable miscible displacements in Hele-Shaw cells based on the three-dimensional, variable viscosity Navier-Stokes equations coupled to a convection-diffusion equation for the concentration field. The simulations exhibit the formation of individual, quasisteady fingers whose properties are characterized as a function of the viscosity ratio and the Peclet number. We observe both traditional tip splitting events, as well as a novel inner splitting mechanism that has not yet been reported in the literature. This tip splitting is associated with fluid transport perpendicular to the plane of the Hele-Shaw cell, and hence cannot be reproduced by gap-averaged approaches. It has the effect of splitting the trailing sections of the finger longitudinally, while the finger tip can largely remain intact.

Tuesday, November 23, 2010 3:05PM - 4:36PM

Session R Z Waves III Hyatt Regency Long Beach Regency F

3:05PM RZ.00001 On weakly nonlinear gravity-capillary solitary waves. BOGUK KIM. As a weakly nonlinear model equations system for gravity-capillary solitary waves on the surface of a potential flow, a cubic-order truncation model is presented, which is derived from the Taylor series expansion of the Dirichlet-Neumann operator (DNO) for the free boundary conditions of the Euler equations in terms of Zakharov’s canonical variables. In deep water, the cubic-order truncation model allows gravity-capillary solitary wave packets in the weakly nonlinear and narrow bandwidth regime where the classical nonlinear Schrödinger (NLS) equation governs. Since this model is consistent to the original full Euler equations in the order of nonlinearity up to the third order, the properties of the gravity-capillary solitary waves of this model precisely agree with the counterparts of the Euler equations. From this cubic order truncation model, the leading-order initial long-wave transverse instability growth rate of the gravity-capillary solitary waves is estimated to be identical, in the weakly nonlinear limit, to the earlier result by Kim and Akylas (J. Eng. Math. 58:167-175, 2007), through an equivalent perturbation transition is sufficiently abrupt that no gradual transition has been observed. When there is no metal ion in the less viscous solution (non-reactive case), the transition was never observed. These results are similar to those obtained in a Hele-Shaw experiment using an associating polymer solution (Zhao & Maher, Phys, Rev, E, 47, 4728, (1993)). We have measured the rheological property of the gel by means of a rheometer and investigated the relationship between the observed fingering-fracturing transition and the measured rheological property. Finally, we discuss the similarity between the present result and the result obtained by Zhao & Maher.

3:18PM RZ.00002 Radiation and diffraction of surface waves by an oscillating water column. RAMIRO GODOY-DIANA, PABLO COBELLI, STÉPHANE RAKOTO-ANDRIANTISILAVO, PAMMM UMR7636 CNRS; ESPCI ParisTech; UPMC; Université Paris Diderot. Radiation and diffraction of surface waves by an oscillating water column. RAMIRO GODOY-DIANA, PABLO COBELLI, STÉPHANE RAKOTO-ANDRIANTISILAVO, PAMMM UMR7636 CNRS; ESPCI ParisTech; UPMC; Université Paris Diderot — An oscillating water column (OWC) modelling a wave energy converter is studied in a small scale laboratory experiment. The system consists of a cylindrical duct partially submerged in a wave tank. The water surface elevation inside the duct oscillates in response to the forcing imposed by an external wave field. The oscillation amplitude inside the duct is maximized when a resonance condition is attained. From a wave energy conversion point of view, the wave field. The oscillation amplitude inside the duct is maximized when a resonance condition is attained. From a wave energy conversion point of view, the
δ

front speed is essentially independent of the interfacial thickness, experimental and numerical data. Measured front speeds show positive agreement with analogue model predictions, which remain strictly single-valued. In contrast to previous studies, we parameterize the amplitude of the interfacial disturbance; the accuracy of this approach is confirmed by comparison against Canada — An investigation of gravity currents propagating through a two-layer stratified ambient of finite vertical extent is presented. Our theoretical discussion Education in China via project IRT0844 and NSFC project 10625210 and Shanghai Sci and Tech. Com. Project 08ZZ43 that the model can be used efficiently to simulate wave propagations in various situations. Acknowledgement: This research is supported in part by Ministry of
errors, we removed the second-order dissipation term and the third-order dispersion term by employing the moments up to fourth order in the lattice Boltzmann methods. Here, we focus on the lattice Boltzmann model for wave equations. Firstly, in order to obtain wave equations with higher-order accuracy of truncation an alternative approach to the traditional methods in computational fluid dynamics. It possesses certain advantages in solving many problems over conventional
for the progressive wave case, effects of wave strain field and Stokes drift are examined. It is found that turbulent Reynolds stress is strongly dependent on the wave phase. Wave normal production, pressure-strain correlation, and pressure transport are essential in the Reynolds stress budget. Vortices are turned, stretched, and compressed periodically by the wave strain field, leading to their wave-phase dependent distribution. Lagrangian average shows that both the Stokes drift and the high correlation between wave strain field and turbulence contribute to the turning of vertical vorticity into streamwise direction.

3:44PM RZ.00004 Wave Generation Experiments Using a Cycloidal Turbine1. STEFAN SIEGEL, THOMAS MCLAUGHLIN, US Air Force Academy — We investigate the wave generation performance of a cycloidal turbine for the purpose of converting wave energy to shaft power. Cycloidal turbines consist of one or more hydrofoils that rotate around a central shaft and can be pitched during rotation. In the present investigation, a two-dimensional wave channel of 45cm width, 4.5m length and a water depth of 30 cm is used. It features a flap wave maker at one end, and a beach at the other end. A two blade Cycloidal turbine model is placed in the center of the wave channel and the generated waves in both up-wave and down-wave directions are measured using wave gauges. We compare the results to inviscid potential flow simulations that show negligible waves traveling up-wave, and a single harmonic wave traveling down-wave making the Cycloidal turbine an ideal wave energy converter if synchronized to the incoming wave.

3:57PM RZ.00005 Simulation of turbulence interacting with free surface and wave . XIN GUO, LIAN SHEN, Johns Hopkins University — Direct numerical simulation is performed for homogeneous turbulence interacting with deformable free surfaces and progressive waves, respectively. For the free surface case, various Froude and Weber numbers are considered. Surface manifestations of the underlying turbulence in the forms of propagating waves and surface roughness are elucidated. Effects of splats and anti-splats on turbulence kinetic energy budget are quantified. For the progressive wave case, effects of wave strain field and Stokes drift are examined. It is found that turbulent Reynolds stress is strongly dependent on the wave phase. Wave normal production, pressure-strain correlation, and pressure transport are essential in the Reynolds stress budget. Vortices are turned, stretched, and compressed periodically by the wave strain field, leading to their wave-phase dependent distribution. Lagrangian average shows that both the Stokes drift and the high correlation between wave strain field and turbulence contribute to the turning of vertical vorticity into streamwise direction.

4:10PM RZ.00006 Lattice Boltzmann Simulations for Wave Propagation. XIUBO SHI, YUEHONG QIAN, Institute of Applied Math and Mechanics, Shanghai University — In the past two decades, the lattice Boltzmann method(LBM) has attracted much attention as an alternative approach to the traditional methods in computational fluid dynamics. It possesses certain advantages in solving many problems over conventional methods. Here, we focus on the lattice Boltzmann model for wave equations. Firstly, in order to obtain wave equations with higher-order accuracy of truncation errors, we removed the second-order dissipation term and the third-order dispersion term by employing the moments up to fourth order in the lattice Boltzmann models with the classical Chapman-Enskog expansion. The time reversibility seems due to the accurate mimicking of the wave equations up to 4th order, that is the absences of the second-order dissipation term and the third-order dispersion term. Secondly, the numerical verification for the model have been carried out, some classical examples are simulated, including wave interference, diffraction, and wave passing through a convex lens. The numerical results demonstrate that the model can be used efficiently to simulate wave propagations in various situations. Acknowledgement: This research is supported in part by Ministry of Education in China via project IRT0844 and NSFC project 10625210 and Shanghai Sci and Tech. Com. Project 08ZZ43

4:23PM RZ.00007 Gravity currents in two-layer stratified media. A.W. TAN, M.R. FLYNN, University of Alberta, Canada — An investigation of gravity currents propagating through a two-layer stratified ambient of finite vertical extent is presented. Our theoretical discussion considers slumping, supercritical gravity currents, i.e. those that generate an interfacial disturbance whose speed of propagation matches the front speed. In contrast to previous studies, we parameterize the amplitude of the interfacial disturbance; the accuracy of this approach is confirmed by comparison against experimental and numerical data. Measured front speeds show positive agreement with analogue model predictions, which remain strictly single-valued. The front speed is essentially independent of the interfacial thickness, δ, even in the limiting case where δ = H so that the environment is comprised of a uniformly stratified ambient with no readily discernible upper or lower ambient layer. Our experiments also consider the horizontal distance, X, at which the front begins to decelerate revealing a non-monotonic dependence on the ambient interface height.

1Supported by the ONR through grant N00014-05-1-0121.

1This material is based upon activities supported by the National Science Foundation under Agreement No ECCS-0801614.